Universidade Federal de Uberlândia
Faculdade de Engenharia Química
Graduação em Engenharia de Alimentos

APPLICATION OF COMPUTATIONAL FLUID DYNAMICS (CFD) IN REFRIGERATING CHAMBERS: REVIEW AND PERSPECTIVES

Patos de Minas - MG
2021

TULLIA MIDORI SEBASTIANI

APPLICATION OF COMPUTATIONAL FLUID DYNAMICS (CFD) IN REFRIGERATING CHAMBERS: REVIEW AND PERSPECTIVES

Projeto de pesquisa apresentado para disciplina de Trabalho de Conclusão de Curso do curso de graduação em Engenharia de Alimentos da Faculdade de Engenharia Química da Universidade Federal de Uberlândia.

Orientador: Prof. Dr. Danylo de Oliveira Silva

Patos de Minas - MG
2021

UNIVERSIDADE FEDERAL DE UBERLÂNDIA
Faculdade de Engenharia Química
Av. João Naves de Ávila, 2121, Bloco 1K - Bairro Santa Mônica, Uberlândia-MG, CEP 38400-902
Telefone: (34) 3239-4285 - secdireq@feq.ufu.br - www.feq.ufu.br

HOMOLOGAÇÃO Nº 46

TULLIA MIDORI SEBASTIANI

Application of Computational Fluid Dynamics (CFD) in Refrigerating Chambers: Review and Perspectives

Trabalho de Conclusão de Curso aprovado nesta data para obtenção do título de Bacharel em Engenharia de Alimentos da Universidade Federal de Uberlândia (UFU) - *campus* Patos de Minas (MG) pela banca examinadora constituída por:

Prof. Dr. Danylo de Oliveira Silva
Orientador - FEQUI-UFU

Prof.ª Dr.ª Yanne Novais Kyriakidis
FEMEC-UFU

MSc. Grégori Ullmann
Doutorando PPGEQ-UFU

Patos de Minas, 15 de julho de 2021.

Documento assinado eletronicamente por **Danylo de Oliveira Silva**, **Presidente**, em 15/07/2021, às 15:29, conforme horário oficial de Brasília, com fundamento no art. 6º, § 1º, do Decreto nº 8.539, de 8 de outubro de 2015.

Documento assinado eletronicamente por **Yanne Novais Kyriakidis**, **Usuário Externo**, em 15/07/2021, às 15:29, conforme horário oficial de Brasília, com fundamento no art. 6º, § 1º, do Decreto nº 8.539, de 8 de outubro de 2015.

Documento assinado eletronicamente por **Grégori Ullmann**, **Usuário Externo**, em 15/07/2021, às 15:30, conforme horário oficial de Brasília, com fundamento no art. 6º, § 1º, do Decreto nº 8.539, de 8 de outubro de 2015.

A autenticidade deste documento pode ser conferida no site https://www.sei.ufu.br/sei/controlador_externo.php?acao=documento_conferir&id_orgao_acesso_externo=0, informando o código verificador **2900141** e o código CRC **EAA23B8B**.

Referência: Processo nº 23117.085948/2019-33 SEI nº 2900141

AGRADECIMENTOS

Primeiramente gostaria de agradecer os meus pais Nelly e Ovídio, e meu irmão Douglas que sempre me apoiaram, me ajudaram em todas as minhas escolhas e decisões e também me possibilitaram fazer essa graduação em Engenharia de Alimentos.

Gostaria de agradecer os meus demais familiares que estiverem presente em toda a minha trajetória.

A todos os meus amigos que conheci antes e durante a minha graduação muito obrigada por estarem sempre ao meu lado e me me proporcionar tantos momentos.

Todas as pessoas que eu pude conviver com a minha passagem pelo PET e pela Atlética, obrigada por agregaram tanto na minha vida pessoal e na minha graduação. Em especial as meninas nas quais eu pude dividir quadra me proporcionando tantos momentos de felicidade e lazer.

Vocês tornaram a minha caminhada mais leve e mais prazerosa.

Gostaria de agradecer em especial a Lela e a Bá que estão comigo desde crianças, a Keke que tornou minha companheira até na peteca, a Cla que conheci na graduação e desde então sempre está do meu lado e o Vincius que me ajuda e me apoia em todos os momentos.

Por fim, mas não menos importante, gostaria de agradecer o meu orientador e tutor do PET, Prof. Dr. Danylo de Oliveira Silva, por me ensinar muito ao longo da minha graduação e me ajudar com o meu TCC.

Todos vocês me ajudaram a crescer ainda mais e chegar aonde eu estou, muito obrigada!

ABSTRACT

Advances in the food sector allows people to have at their home a variety of food, but for this to happen, in a country as large as Brazil, sometimes the food needs to be transported a long way in a road by a truck. The cold chain, is used to maintain the quality and increase the shelf life of perishables foods. In order to keep the cold chain, the trucks need to maintain the optimal temperature throughout the transportation. Therefore, the aim of this study was to make a literature review of the use of Computational Fluid Dynamics (CFD) to simulate a refrigerated truck. CFD simulations can be performed to solve many refrigeration problems, such as to analyze the refrigeration unit, inlet and outlet positions, airflow, temperature, cargo design and air curtains. For this purpose, characteristics of the system and consequently the truck must be well known to determine the governing equations, boundary and initial conditions and others parameters needed. Furthermore, studies showed that software FLUENT (ANSYS, Inc.) is the most used in the simulations of refrigerated trucks. Altogether more studies can be made with loaded trucks, different configurations and including more external factors. Studies showed that the maintainance of cold chain still have deficiencies and the use of CFD can help to improve the positioning of inlet, outlet and air courtains, the refrigeration unit and the cargo configuration inside the truck.

Keywords: Cold chain, truck, simulation, application.

LIST OF FIGURES

SUMMARY

1 INTRODUCTION

Globalization and development of technology in the food sector allow the consumer to choose among a variety of products and change the eating habits towards fresh food with high quality. However, to comply with this market request, most of the time the food needs to be transported to different places. The food supply chain, along with the evolution of the markets, must be agile to satisfy the demand according to quality assurance standards (MEJJAOULI; BABICEANU, 2015; YILDIZ, 2019).

Brazil is the fifth largest country in territorial extension in the world, making even interstate transport exceedingly difficult. According to the Ministry of Infrastructure, the federal highway network has a length of 75.8 thousand km, of which 65.4 thousand km are paved and 10.4 thousand km, or 13.7%, are unpaved highways (BRASIL, 2019). In addition, according to the National Department of Transport Infrastructure (Portuguese abbreviation – DNIT) (2014), the road network has a total extension of 1,691,163.80 km, and only 12% are paved.

Food transportation must be well planned to reduce, as much as possible, the loss of product during the process, mainly because road transport in Brazil is not fast and effective in some parts of the country due to the small portion of paved roads. According to the Food and Agriculture Organization of the United Nations (FAO), 1.3 billion tons of food are wasted or lost along its production chain each year. Therefore, some measures must be taken to reduce the waste along the production chain, such as the use of the cold chain as an alternative.

The cold chain consists of using low temperatures throughout the food processing to delay chemical reactions and enzymatic activity, besides inhibiting the growth of certain microorganisms in the food, that is, it aims to maintain the quality and increase the shelf life of the product (SCHMITZ, 2016). This process is extremely important in the food industry to preserve the quality, especially of perishable products, being necessary to maintain a constant low temperature throughout the chain, including during road transport. To preserve the temperature in road transport, for refrigerated cargo, it is necessary to use trucks with refrigerated bodies (YILDIZ, 2019).

Transport within the cold chain still faces some difficulties in monitoring the refrigerated cargo, as there may be a 30% increase in temperature during the trip from the producer to the Distribution Center (DC), and around 15% on trips from the DC to local retail marketing (SPAGNOL et al., 2018). Thus, it is a great challenge to maintain the cold chain throughout the route from the producer to the consumer without reducing the quality of the product (SPAGNOL et al., 2018). Computer systems can be used to design bodies of trucks, evaluating heat gains, airflow and temperature gradients. Among computational tools, Computational Fluid Dynamics (CFD) techniques are widely used (GETAHUN et al., 2017).

Computational Fluid Dynamics techniques are used for two- or three-dimensional simulations of fluid flow, although its use is not exclusive for refrigeration cases. CFD techniques have been used more and more because they are less expensive than experimental studies and the results are very detailed (NORTON; SUN, 2006). For refrigeration trucks, the techniques can be used to assist in the understanding of airflow during refrigeration taking into account the cargo loss present, being able to detect weaknesses in the process, and apply changes for improvement, such as optimization of cooling speed air (BADIA-MELIS et al., 2018). The transportation and maintenance of the food cold chain still have deficiencies, so the understanding of tools, as CFD techniques, which can improve these activities has a great economic impact.

2 OBJECTIVE

The general objective of the present work is to make a bibliographic review on the use of computational fluid dynamics (CFD) techniques in trucks refrigeration chambers, and the specific objectives are: (i) to know the most used models in this type of simulation, as well as the hypotheses and simplifications adopted; (ii) present future perspectives on the use of the techniques.

3 LITERATURE REVIEW

3.1 Cold chain

Production, storage, and distribution are linked steps significantly important in the productive chain of high-temperature sensible products. In those cases, it is applied the cold chain, which consists of a series of interdependent processes to ensure a low temperature and to improve food conservation, throughout the processing steps until the consumer place the food in a domestic refrigerator (BADIA-MELIS et al., 2018; JARA et al., 2019). This process is used to decrease respiration rate, when necessary, reduce enzymatic activity, slow up or inhibit microbial growth and decrease ethylene production to slow up ripening or food deterioration (HAN et al., 2017). The cold chain is applied in fruits and vegetables to decrease metabolism which results in a faster deterioration after the harvest (SPAGNOL et al., 2018).

About one-third of fresh food is discarded because conservation and sanitary conditions are inadequate (SPAGNOL et al., 2018). For this reason, the maintenance of the cold chain is very important. Even a small fluctuation in temperature, for a certain time, at any stage of the chain, can condemn the effort made to refrigerate previous steps.

The cold chain has an economic importance because by guaranteeing that the food will be transported in an environment with proper temperature during its journey, the shelf life is prolonged and the quality level is kept, thus providing a better performance for interregional and transnational long-distance sales (HAN et al., 2017). However, in order to achieve an effective conservation of food in the cold chain, the refrigerated truck must be correctly designed to provide an excellent performance without great temperature variation during transport (ZHAO; ZHANG; XU, 2020).

Due to developments in the refrigerated transport sector, the tolerance for temperature variations is specified as ±2ºC. Fruits and vegetables are more sensible during transport, due to the production of heat, humidity, and fast degradation due to their transpiration and ethylene production. For this reason, ventilation inside containers with those products must be appropriate to conserve, avoid microbial growth, change of color, and loss of firmness (YILDIZ, 2019).

3.2 Refrigerated trucks

During the transportation of certain foods, it is necessary to control aspects such as time, room temperature, and risk of deterioration. For products that require a lower temperature during processing, refrigerated trucks are used to keep the temperature in an ideal range. Stages between the beginning of production and the consumer in a food production chain are linked with refrigerated transport, reinforcing the importance to keep the ideal temperature in all stages to maintain the quality to the consumer. The vehicle must maintain the ideal temperature, be in a good hygienic condition, and do not contain contaminants. The reduced size of the chambers keeps the temperature constant, and external insulating paints are also suitable (SCHMITZ, 2016; LIU; SAMAN; BRUNO, 2012; JARA et al., 2019).

The need for refrigeration equipment depends on the type of the truck body and its application, being defined through thermal exchange capacity. So, it’s necessary to calculate the thermal load needed to the body, that means, how much heat must be removed from the internal environment to keep an appropriate temperature during the operating time established (SCHMITZ, 2016).

According to Zhao, Zhang and Xu (2020), refrigerated trucks have wide temperature control, carry a diverse volume of goods, can be used for long-distance transportation, and they differ from a common refrigeration body only because they are airtight and have thermal insulation performance. However, most refrigeration systems can not extract heat from the product, they only maintain the temperature according to the product specification (JAMES, 2019). These trucks are mainly used to transport cold drinks, eggs, meat products, fruits and vegetables, and even in sectors like mining, military, and health (ZHAO; ZHANG; XU, 2020).

A refrigerated truck can be divided into three main parts: the chassis of the vehicle; a body; and the refrigeration unit. Besides, it can have a single body or more, depending on the demand and according to the quantity of product to be transported. The body structure is generally composed of three layers with good sealing performance and thermal insulation to ensure appropriate refrigeration (ZHAO; ZHANG; XU, 2020).

Some variables must be considered in the transportation of goods such as, heat sources air circulation conditions, thermal insulation characteristics of the transport equipment, pre-cooling and temperature requirements of the product transported, characteristics of the products, products respiration rate, package design, package arrangement, and the possible existence of mixed cargo (BAPTISTA, 2006; GETAHUN et al., 2017). When the products loaded are already at the desired temperature, and if they don't generate heat, the cargo needs only to be isolated from external environment and surrounded by a blanket of cooled air to maintain the appropriate temperature. The contact of the product with the internal surface of the vehicle must be avoided while loading the truck as it allows the exchange of heat between them (JAMES, 2019).

These trucks can be used to transport refrigerated or frozen food. Chilled food has a temperature above the freezing point, so the controller must be regulated to avoid reaching the freezing temperature. On the other hand, frozen food is kept at a temperature between -10 ºC and -25ºC, and the controller must be regulated to avoid temperatures higher than the freezing point (BAPTISTA, 2006).

3.2.1 Thermal insulation

The insulation material and the body structure directly affect the thermal insulation, the mechanical resistance, the performance regarding the environment and the energy comsumption economy of the truck (ZHAO; ZHANG; XU, 2020). The thermal insulation is extremely important to minimize the heat transfer between the external and internal environment, and decrease energy consumption and CO_2 emission but to be effective, the insulation materials must be periodically tested to assess whether they are within the specifications because these materials deteriorate over time with a 3 to 5% estimated loss per year (JAMES, 2019; ARTUSO et al., 2019). According to Artuso et al. (2019), insulation can be classified in two categories according to the coefficient of heat transfer of the insulation: Reinforced Insulation (IR) or Normal Insulation (IN), with values below 0.4 $W \cdot m^{-2} \cdot K^{-1}$ and between 0.4 and 0.7 $W \cdot m^{-2} \cdot K^{-1}$,respectivelly.

Expanded polyurethane is a widely used insulation material, it can be used alone as Rai, Sun and Tassou (2019) and Han et al. (2016) studied, or combined with another insulation material. Getahun et al. (2017) used an isopanel constructed with a layer of polyurethane between two metal sheets, and Artuso et al. (2019) studied three layers of polyurethane foam between two thin fiberglass-based layers. The type of insulation depends on the conditions of the system, but the insulation must be compatible and must not interact with the product being transported. Thereby, Jara et al. (2019) studied the thermal behavior of a refrigerated vehicle with a body composed of sandwich panels insulation made of galvanized sheet in the outer, a food grade stainless sheet in the inner and expanded polyurethane in between them, to a refrigerated truck for food transportation.

Other materials are being studied such as vacuum insulation (VIP) by Verma and Singh (2019, 2020) and Lakatos and Kovács (2021), phase change materials by Fioretti, Principi and Copertaro (2016), airgel insulation and mineral wool. The VIPs efficiency can be up to five times greater than the panels with insulated foam, and also a smaller thickness is needed, increasing the space for the cargo. However, this technology has a problem in the insulation of the corners and joints. Currently, an internal and external metal sheet is the most used material for insulation, but if it is substituted by a lighter and less conductive material it would be possible to improve energy consumption and efficiency (JAMES, 2019).

3.2.2 Refrigeration systems

The most used system in refrigerated trucks is the conventional mechanical refrigeration system, but other alternatives are being developed. The energy efficiency rate (EER), also known as the coefficient of performance (COP), is used to evaluate the different systems. This parameter compares alternatives from different power sources, calculates the associated costs, environmental impacts and it is defined by Equation 1 (JAMES, 2019).

$$EER = COP = \frac{Q_{cool}}{|P_w|} \leq \frac{T_{cold}}{T_{hot} - T_{cold}} \tag{1}$$

Being Q_{cool} the heat absorbed from the body to be cooled, in Joules, and P_w the energy absorbed to reduce the body temperature. Considering the ideal Carnot cycle, the relation can be made depending on the temperatures.

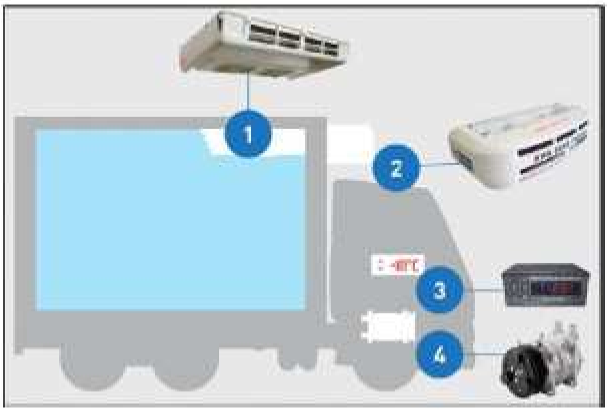
3.2.2.1 Mechanical system

The mechanical refrigeration system uses a traditional heat transfer mechanism and consists of removing heat from the internal environment and releasing it in an external environment through a refrigerant-based on a non-spontaneous process (BERNSTEIN, 2017). Usually, the truck refrigeration system consists of mechanical refrigeration through vapors, which allows the continuous circulation of the coolant (SCHMITZ, 2016). Thus, according to Baptista (2006) the principle of mechanical refrigeration of the body is composed of four parts:

- compression: in the compressor, the refrigerant gas is compressed, i.e., the pressure and temperature of the gas are increased. Then the gas at the high pressure is discharged into the condenser;
- condensation: in the condenser, the gas with a high temperature and pressure is cooled by air or water. The gas changes into a liquid phase, still at high pressure;
- expansion: the expansion valve controls the flow of the refrigerant, which is received from the condenser (possibly through a liquid receiver), so the correct amount of the coolant goes through the evaporator;
- evaporation: when entering the evaporation section, the refrigerant passes from the side with the highest pressure, through a small role in the expansion valve, to the side with the lowest pressure of the system. The lower pressure causes it to evaporate. The latent heat of evaporation is extracted from the coolant, for example, through the passage of air in the evaporation coil. Then the refrigerant gas returns to the compressor and the cycle is repeated.

The mechanical refrigeration units can be driven by an electric motor, or an independent one, as shown in Figure 1. The most common is the one that has an independent plug unit set up in an opening provided in the front wall of the vehicle. The condenser is located on the outside and the evaporator inside the unit. The compressor can be triggered by a belt or, as usually, directly triggered by an auxiliary engine that uses the gas from the trucks' tank or an independent one (JAMES, 2019).

Figure 1 Mechanical refrigeration system: (1) evaporator. (2) condenser. (3) control panel, (4) compressor.



Source: Jara (2019).

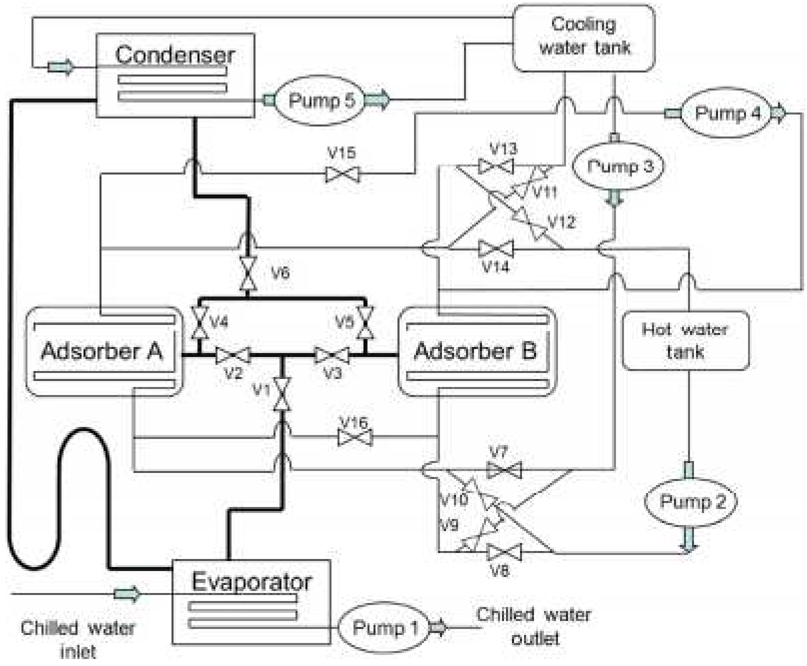
Most long-distance road transport is mechanically cooled. However, there is still great difficulty in maintaining the temperature of perishable foods when the truck delivers to several establishments per trip. So, the refrigeration system must be designed to allow the removal of products and different load distribution during the deliveries and the days of the week (JAMES, 2019).

3.2.2.2 Adsorption systems

The search for green refrigeration is increasing, which is why studies are being carried out to replace the mechanical refrigeration system. The refrigerant used in mechanical refrigeration system contributes with global warming and ozone depletion, thereby adsorption systems are attractive alternatives because due to the use of natural refrigerants (PAN; PENG; WANG, 2021).

A kind of adsorption refrigeration system is the double-bed adsorptions cooling system, shown in Figure 2, in which consists of two phases. In the first phase adsorption happens in a Adsorber A , and desorption in Adsorber B. While on the second phase desorption takes place in Adsorber A, and adsorption takes place in Adsorber B. Pefore operation the system components and the pipeline for water vapor or condensed water are evacuated, then water is introduced into the evaporator and U-tube and the whole system is filled with low pressure water vapor. One of the adsorbers, Adsorber A, is connected to the evaporator. The water vapor is adsorbed by the silica gel adsorbent in the adsorber, maintaining a low pressure condition inside the evaporator, and thus the water can evaporate at a low temperature. There is a heat generation during the adsorption process in the adsorbent that is removed by providing the cooling water. The desorption phase happens ina a higher pressure environment, the other adsorber, Adsorber B, is heated and water vapor is desorbed from the adsorbent. Due to the pressure difference of the adsorbent and the condenser the desorbed water vapor is transferred to the condenser. Thereat, the condensed water flows back to the evaporator via the U-tube to complete the cycle, connecting the condenser and the evaporator.

Figure 2 Schematic diagram of adsorption cooling systems**.**



Source: Chan et al. (2015).

This system has lower COPs and higher-cost when compared to the mechanical refrigeration system, however, it can use residual heat which improves the overall consumption of the system, has simple control, low operating cost, and less vibration (JAMES, 2019; PAN; PENG; WANG, 2021). According to Gao et al. (2016), the investment in an adsorption system on a refrigerated truck can have the investment returned in less than a year when compared to a mechanical refrigeration system.

3.2.2.3 Phase change materials systems

The first refrigeration systems used naturally produced materials with phase change (PCM), like the ice, and nowadays it can be organic, non-organic, or compound materials. In this method it is used phase change materials, in temperatures below their freezing point, to absorb the heat inside the truck during the transportation, leading to a phase change in the refrigerant. This type of refrigeration system is still used in refrigeration to serve local distribution chains, and transport products as ice cream and can also be combined with other refrigeration systems, use of plates, bags, or boxes with phase change material, to improve the refrigeration (JAMES, 2019; MOUSAZADE; RAFEE; VALIPOUR, 2020; LI et al., 2021).

Phase change materials have a lower impact on the environment and need for energy, due to their capability to store and release a large amount of heat energy reversibly (LI et al., 2021). The phase change system consists of a coil in which the primary refrigerant passes through, assembled inside a thin tank and filled with a PCM, with standard freezing temperatures between 3 and 50ºC. Then plates are fixed on the walls and ceilings or used as shelves and partitions in the vehicle (JAMES, 2019). This method produces no noise and can keep refrigerating even with the engine off, but a relatively high mass of the material is necessary (MOUSAZADE; RAFEE; VALIPOUR, 2020).

If the system is designed to transport frozen food, it cannot be used for chilled food because the temperature cannot be adjusted, so the material has to be chosen correctly to keep the demanded temperature (MOUSAZADE; RAFEE; VALIPOUR, 2020; LI et al., 2021). However, Maiorino et al. (2019) used it coupled with mechanical refrigeration obtaining less temperature fluctuation and reduction of the temperature gradient, and Liu, Saman and Bruno (2012) developed a refrigeration system with phase change material with comparable weight to a conventional refrigeration system and less than half of the energy cost, showing that this type of refrigeration can be managed to be efficiently used. Most of the phase change materials used for cold storage have a low-temperature phase change, showing melting temperatures below 10ºC, due to the high latent heat and constant phase change temperature required (LI et al., 2021).

3.2.2.4 Cryogenic system

Cryogenic refrigeration consists of the use of a cryogenic liquid, that is, substances that at ambient temperature and pressure are in a gaseous state and to be liquefied must be subjected to very low temperatures, such as nitrogen, argon, and carbon dioxide. One kind of structure of the liquid nitrogen system is formed by a liquid nitrogen storage tank connected to a boom sprayer along the roof of the transport vehicle. Nitrogen is released by a thermostatically controlled valve and vaporizes instantly upon entering the body. The air is cooled due to changes in the sensible and latent heat of the nitrogen liquid, and as soon as the desired temperature is reached the valve closes the nitrogen flow by controlling the temperature by intermittent injections. Another cryogenic refrigeration system consists of a Joule-Thomson cryogenic refrigeration system, formed by micro J-T cryocoolers composed of a heat exchanger, throttling element, and an evaporation chamber (JAMES, 2019; GENG et al., 2019).

Transport systems with nitrogen liquid have many advantages, such as minimum maintenance requirements, uniform load temperature, low-noise operation, low capital cost, environmental acceptability, rapid temperature reduction, and increased shelf life due to the creation of a modified atmosphere. (JAMES, 2019).

This system allows long-distance transports, keeping the load cooled to 3ºC for up to 50 hours with only one load of liquid nitrogen. But, the limitation of this type of transport system is related to the uncertainty regarding the cryogenic liquid prices, availability, and lack of loading structure (JAMES, 2019).

3.2.3 Airflow

Even with the use of any of the refrigeration systems, the product needs to be surrounded by air or a surface at the desired temperature or below it, to maintain the ideal temperature. The poor distribution of air causes heterogeneity in the temperature, humidity, and other variable concentrations, and the change in those parameters can contribute to deterioration and loss of the product's quality during transport. For this purpose, it is possible to install an air circulation system, forced or by gravity, during the load (JAMES, 2019; MOUREH; FLICK, 2005).

Conventional forced air circulation systems usually discharge air onto stacked or suspended products directly from the evaporator, or through ducts. The air flows towards the rear cargo doors with relatively high speed to remove, as far as possible, the heat flows generated by the cargo and exchanged by the walls. However, it is important to study the position and dimension of the inlet and outlet airflow to have an efficient ventilation (MOUREH; FLICK, 2004, 2005; JAMES, 2019).

The over-compaction of the cargo, or small spaces between the pallets creates resistance to airflow and results in an uneven distribution of air through the cargo and poorly ventilated areas. These zones are observed at the rear of the vehicle and can cause higher temperature points in the load, even when the cooling capacity is greater than the generated and exchanged heat flows (MOUREH; FLICK, 2004).

There are not many studies of refrigerated trucks with pallets inside and this may be due to the difficulty to directly measure the speed and flow of air in small spaces. Getahun et al. (2017) simulation study shows that the arrangement of the pallets inside the container affects the circulation of the air. In addition, a full-scale experiment carried out showed that the distribution of the airflow inside the container is very uneven, in the front part, opposite the door, with high-pressure gradients due to high speeds and behind with low-pressure gradients (MOUREH; FLICK, 2004). In this context, CFD techniques can be used as a tool for understanding the airflow and temperature distribution and also to improve the refrigeration system of trucks during transport.

3.3 Computational Fluid Dynamics (CFD)

The development of more robust algorithms for the solution of systems of linear algebraic equations and the proposal of more elaborate models to represent dispersed phases, combined with greater computational processing capacity and improvement in data storage capacity, have been allowing the evolution of tools to study fluid dynamics. Among several alternatives, Computational Fluid Dynamics (CFD) stands out, which uses a discrete or punctual solution (discretized equations) instead of continuous solutions of the differential equations involved a the process (SILVA, 2012).

According to Norton and Sun (2006), Computational Fluid Dynamics (CFD) was developed over the years based on studies progress made by Richardson in 1911 and Courant, Friedrichs, and Lewy in 1928 and has undergone advances that allow its application in several areas today. The progress of this tool has been so great that nowadays it is used as much as the traditional didactic methods of experimentation and analytical modeling to solve fluid flow problems. Its advantage concerning experimental methods refers to the lower cost and time, in addition to producing efficient and detailed results (NORTON; SUN, 2006).

CFD techniques have become essential tools in fluid dynamic studies for allowing to carry out a deep analysis of the fluid and the effects in a lot of equipment. Some advantages of using CFD techniques include: (i) the details provided to understand flow distribution, weight losses, mass, and heat transfer; (ii) the possibility to evaluate geometric changes faster and with less costs than laboratory testing; (iii) the capability of answer many ‘what if’ questions in a short time depending on the studied physics; (iv) the possibility to reduce scale-up problems; (v) the simulation of conditions that are dangerous to make measurements, such as high temperatures; (vi) the ability to highlight the root cause besides evaluating the problem. Although, it is important to emphasize that experimental data are frequently used to validate CFD models through comparing their results. (SADREHAGHIGHI, 2019; JARA et al., 2019).

In the food industry, the use of CFD is increasing, and some areas of application are homogenization, drying, cooking, sterilization, refrigeration, and cold storage. The main objective of the application studies is to increase the efficiency in the processing and the storage steps, allowing to attend the market that is expanding and requiring even more quality of the products (NORTON; SUN, 2006).

3.3.1 Steps of CFD analysis with commercial softwares

CFD analysis consists of codes with numerical algorithms that can solve a fluid flow problem. The process of analyzing a phenomenon via CFD includes three phases that are performed in specific software: pre-processing performed in CAD and mesh generation (SolidWorks, AutoCad, Gambit, Meshing, Pointwise and others), solvers (Fluent, CFX, StarCCM and others) and postprocessor (Fluent, CFX, CFDpost, TecPlot and others) softwares. In the preprocessor the definition of the model is carried out and it is the most important phase, because the simulation depends on the restrictions and conditions inserted in this first environment. The solver, second environment, receives the previously entered data, structures them in a soluble arrangement, and resolves by interactive methods considering the imposed boundary solutions. Finally, in the third environment, the post-processor is used to visualize the solution field. In some cases, the techniques offered by the software do not fit the problem, so solution files can be imported from spreadsheets or other visualization programs (NORTON; SUN, 2007; VERSTEEG; MALALASEKERA, 2007).

3.3.1.1 Preprocessor

In the first environment, the data of the fluid problem are inserted in the program interface for later use by the solver. In this way, the preprocessor is linked to the success of the problem solution, because it has all the untreated data and mathematical statements that are going to be used in the modeling (NORTON; SUN, 2007; VERSTEEG; MALALASEKERA, 2007).

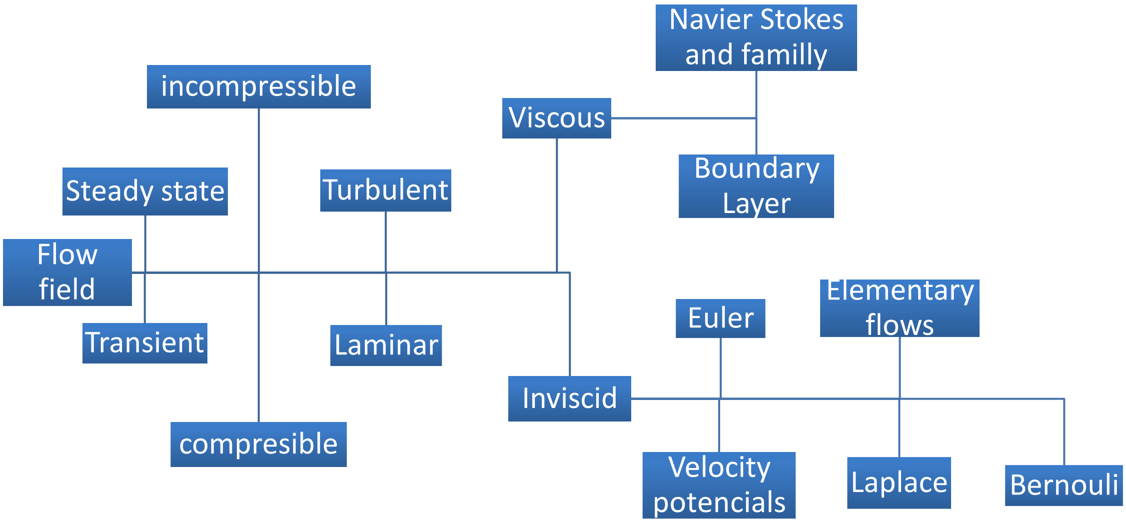
Initially, in this environment, the geometry of the region of interest is defined, that is, the computational domain. To choose the correct domain it is important to understand, in details, the physics of the problem and factors that could influence the parameter analyzed. The geometry can be chosen through a software database with previously designed structures or more commonly created or imported to any standard Computer Assisted Design (CAD) program (NORTON; SUN, 2007; VERSTEEG; MALALASEKERA, 2007; ZHAO et al., 2016).

The next step is to generate the mesh, which consists of subdividing the domain into smaller elements, without overlapping. Usually, the ideal grids are not uniform, being refined in areas with greater variation of responses from point to point, such as in the boundary layer, and coarser in other regions. One factor that can define the solution of the problem is the knots within each cell, and the precision, cost, and time required are governed by the number of cells in the mesh. In other words, the more cells, the more accurate the solution will be, but higher will be the cost of the hardware and the time required for the solution, so it is important to balance calculation accuracy and computational cost. Although, the increase in the number of cells reaches a point of stagnation, where the numerical solution is independent of the mesh size configurating the optimal point. Computational time and simulation costs can be decreased by exploiting symmetries or periodicity, thereby, reducing the domain. It takes a lot of effort and time to obtain a good quality mesh, so the major codes can import data from mesh generators, such as PATRAN and I-DEAS (ZHAO et al., 2016; NORTON; SUN, 2007; VERSTEEG; MALALASEKERA, 2007). Commercial CFD packages, most of them, include programs to simultaneously define the domain and construct the mesh (ZHAO et al., 2016).

The domain geometry and grid design specifications are important tasks in the pre-processing. Later, the user needs to obtain a satisfactory simulation solution, in which the convergence and independence of the grid are the aspects that characterize these results. In a convergent solution, the residues, measures of general conservation of the flow properties, are very small and the solution's algorithm is iterative. A careful selection of settings for relaxation factors and acceleration devices assists in the search for a convergent solution. Optimizing the speed of the solution requires experience with the code itself and good initial grid design, and depends on the knowledge of the expected flow properties. The only way to eliminate grid errors is to carry out a grid dependency study, in which an initially coarse grid goes through successive refinements until there is no change in some points of the result, and makes the simulation independent of the grid. In high-quality CFD studies, the search for the independence of the grid results is essential (VERSTEEG; MALALASEKERA, 2007).

Lastly, there is the selection of the model that represents the chemical and physical phenomena, the determination of fluid properties, and boundary conditions. This step requires knowledge of the initial status of the model and the potential solution since it has a great influence on the efficiency and accuracy of the calculations These conditions determine if a well posed problem will be obtained, meaning that the solution, which means that the solution of the problem exists, is unique, and depends continuously on the data (NORTON; SUN, 2007; SADREHAGHIGHI, 2019; VERSTEEG; MALALASEKERA, 2007; ZHAO et al., 2016). Sandrehaghighi (2019) made a road map of the physics set-up for CFD and it is shown in Figure 3, that was adapted by the author.

Figure 3 Road map for CFD studies.



Source: adapted from Sadrehaghighi (2019).

About 50% of the time spent to develop a CFD project consists of defining the geometric domain and generating the mesh (ZHAO et al., 2016). In addition, the preprocessor gives access to material-owned databases of common fluids and the facility to acquire models of physical and chemical processes with the main fluid flow equations. The math modeling in this step consists of defining the physical problem, creating a mathematical partial differential equation (PDEs) model (systems of PDEs, ordinary differential equations – ODEs –, and algebraic equations), defining initial and boundary conditions to get a well-posed problem, creating a discrete (numerical) model, discretizing the domain, generating the grid and obtaining a discrete model (SADREHAGHIGHI, 2019; NORTON; SUN, 2007; VERSTEEG; MALALASEKERA, 2007).

3.3.1.2 Solver

The solver, together with the CFD software, organizes the mathematical data inserted in the preprocessor in numerical matrices and solves them through an iterative method (NORTON; SUN, 2007). According to Versteeg and Malalasekera (2007), there are three distinct lines of numerical solution techniques: finite-difference; finite-element; and finite-volume. In engineering fields, the finite-difference techniques are rarely used, finite-element is used specially with complex geometrical structures, and finite-volume is the most popular because it is easy to understand, to program, and has a high computational efficiency (ZHAO et al., 2016). In general, these numerical algorithms have the following steps: integration of governing fluid flow equations over all finite control volumes in the domain; discretization, which consists of converting integral equations into a system of algebraic equations; solution of algebraic equations by iterative method (VERSTEEG; MALALASEKERA, 2007; ZHAO et al., 2016).

The finite volume method, a finite difference formulation, is part of the most well-established CFD codes (CFX / ANSYS, FLUENT, PHOENICS, and STAR-CD). This method differs from other CFD techniques on the integration of the volume control because the results express the exact conservation of properties relevant to each finite cell size. The main attraction of the method is the clear relationship between the numerical algorithm and the principle of implicit physical conservation, which makes the concepts simpler to understand than in finite element and spectral methods (VERSTEEG; MALALASEKERA, 2007).

Iterative methods are usually used to solve a set of discretized equations to be applied to a single dependent variable. Some methods used in CFD that have become common techniques used in many commercial packages are: TDMA solutions (tri-diagonal matrix algorithm), solving line by line the algebraic equations; segregated solver, the semi-implicit method for pressure-related equations (SIMPLE), used by most commercial CFD packages; and their descendants, such as SIMPLEST, SIMPLER, SIMPLEC and PISO (ZHAO et al., 2016; NORTON; SUN, 2007; VERSTEEG; MALALASEKERA, 2007). These methods close the discretized moment equations with the continuity equations, sequentially, determining the pressure field. The increase in the number of cells turns deeper the elliptical nature of the pressure field and consequently decreases the rate of convergence considerably. As a result, it was created a multigrid technique that computes speed and pressure corrections simultaneously, thus increasing convergence rates. Many CFD packages, even those based on unstructured meshes, can employ multiple meshes as a standard solver option (NORTON; SUN, 2007).

Therefore, this environment organizes into numerical arrays and solves with iterative methods the input from the preprocessor. Usually, this step requires to solve a huge number of equations. Before initiate the solution the type of simulation concerning time must be chosen: steady state or transient simulation.. A steady-state simulation is used to simulate conditions in dynamic balance, while transient simulation is used to evaluate a parameter over time (ZHAO et al., 2016).

3.3.1.3 Postprocessor

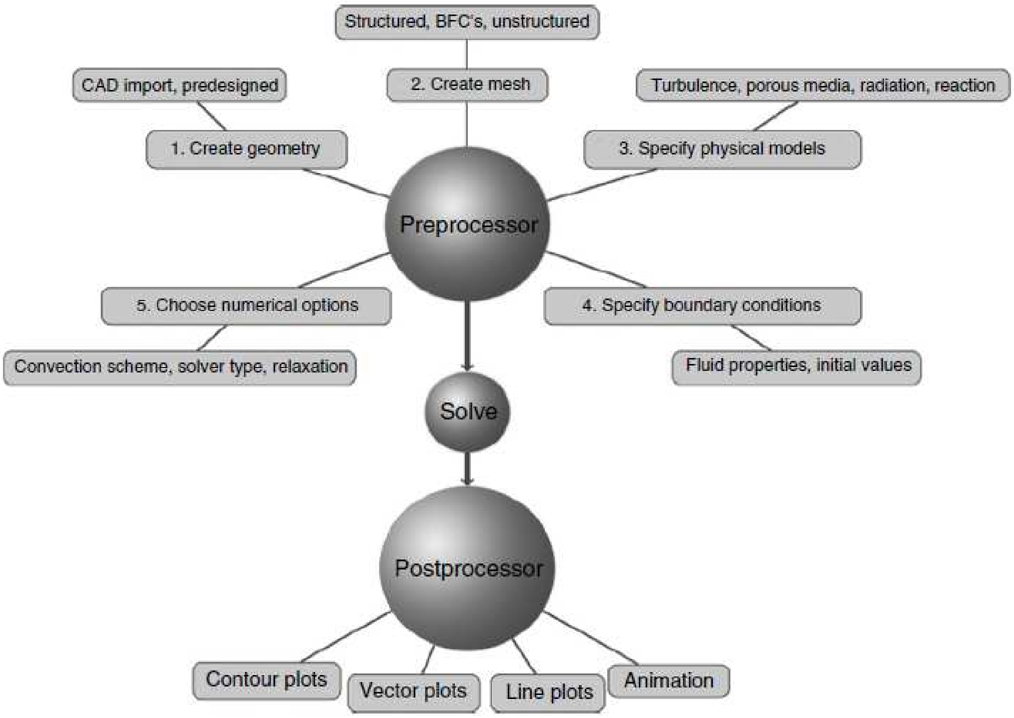
When the solving process is completed, the simulation results are generated. The third environment allows the user to view and examine the resulting solution, with domain geometry, grid display, vector graphics, shaded and linear contour graphics, 2D and 3D plots, particle tracking, manipulation visualization, and animations. These tools improve the interpretation of the results and are being progressively improved in commercial software packages. In addition, some packages allow exporting data fields to external modeling programs, enabling performing other subsequent processes. The graphics output features of CFD codes, as well as in other branches of CAE (Computer-Aided Engineering), have revolutionized the communication of ideas to non-specialists. To comprehensive evaluate a simulation accuracy, authenticity and satisfaction, the postprocessing is essential (ZHAO et al., 2016; NORTON; SUN, 2007; VERSTEEG; MALALASEKERA, 2007).

Before executing the simulation there is a step of identification and formulation of the flow problem in terms of the physical and chemical phenomena that need to be considered. So, while solving a fluid flow problem, it is important to be aware that the physics and chemistry involved are complex and the results generated by a CFD code are directly linked to the data embedded in it. Thus, the user of a code must be competent in several areas to obtain a quality result (VERSTEEG; MALALASEKERA, 2007). According to Versteeg and Malalasekera (2007), some decisions that must be taken are: if the modeling will be carried out in two or three dimensions; if the effects of ambient temperature or pressure variations in the density of the airflow will be excluded; solve the turbulent flow equation or neglect the effects of small air bubbles dissolved in the water. To reduce the complexity of the problem, it is necessary to use simplifications and it affects the quality of the information generated by CFD, so it is essential to have modeling skills and to be aware of all the assumptions made (VERSTEEG; MALALASEKERA, 2007).

Each numerical algorithm has its characteristic errors and some terms such as numerical diffusion, false diffusion, and numerical flow can be used to describe them. The user must analyze whether the result is the expected only with a deep knowledge of the algorithm, with this it is possible to recognize possible standard errors. A to assess the validity of the model or the accuracy of the final results is by comparison with an experimental test. Therefore, the user must know that the CFD is not a substitute for experimentation, but rather a powerful additional tool for solving problems. For the validation of a CFD code, highly detailed information about the boundary conditions is essential and it generates a large volume of results, besides producing experimental data with a similar scope to the simulated one. When it is not possible to carry out the validation in an experimental way, the user should analyze previous experiences, compare with analytical solutions of similar simpler flows and compare with high-quality data of related problems, reported in the literature (VERSTEEG; MALALASEKERA, 2007).

In this way, CFD involves the creation of a set of numbers that constitutes a realistic approximation of a system with a high-level detailed result. But, to obtain a high-quality result, it is essential to validate the result rigorously and frequently to have complete experience and understanding of the physics of the fluids and the fundamentals of numerical algorithms. Figure 4 shows a resume of all the environments and steps to realize a CFD simulation (VERSTEEG; MALALASEKERA, 2007; NORTON; SUN, 2007).

Figure 4 Environments and steps of CFD simulation.



Source: Norton; Sun (2007).

3.3.2 Commercial CFD codes

To increase the application of modeling in some areas of research, CFD codes have undergone great development. However, this development was not uniform, and the offered functionalities vary between codes due to competition between developers. Food engineering requires some qualifications, such as the ability to import mesh, boundary and initial conditions, modeling of non-Newtonian fluids, two-phase flow, flow-dependent properties, the start of phase change, and flow through porous media. Not all codes can be used because it does not have the necessary functions, so it is important to evaluate the functional considerations of the code for the correct choice. However, some commercial softwares allows the user to implement

User Defined Functions (UDFs), that are customized functions to atend different situations from those that the software brings. The CFD codes most commonly used involves CFX, ANSYS FLUENT, PHOENICS, STAR-CD, COMSOL, STAR-CCM+, FLOW-3D, and AUTODESK CFD (LONG et al., 2021; NORTON; SUN, 2007; FLUENT ANSYS, 2013).

3.3.3 Use of CFD in refrigerated trucks

One of the main segments of the cold chain is the refrigerated container, besides the pre-cooling, storage, handling in distribution centers, and refrigerated display. To preserve the quality of the product and reduce losses, the cooling must be uniform inside the truck so that the temperature is the same in all products and there is no increase in the temperature. Temperature changes can happen due to the production of heat by the commodity and other sources of heat, such as fan motors, heat infiltration from the outside, solar radiation, among others, leading to quality loss (GETAHUN et al., 2017).

The system of cold chain maintenance is complex because it involves many variables that make it difficult to carry out the project on an experimental basis. In the last decades, numerical models have been validated, which obtain important information about the distribution of airflow and temperature within cold storage systems. CFD techniques are the most chosen methods due to their great visualization capacity, high temporal and spatial resolution of airflow, and temperature distribution with an acceptable level of precision (GETAHUN et al., 2017).

CFD techniques are not a process for the cold chain itself, but it helps understanding the airflow pattern during the cooling process or maintenance of low temperatures (BADIA-MELIS et al., 2018). This tool allows evaluating points to be improved and to apply changes such as optimize the speed of forced air for refrigeration, creating an improved ventilation system for product boxes, adjusting the distribution of pallets during the cooling process, ensuring uniform cooling during transportation and storage, and reducing energy consumption in the pre-cooling stages (HAN et al., 2017).

The use of CFD for numerical predictions of airspeed and temperature can be obtained by solving sets of differential equations of mass, moment, and energy, written in their conservative form using the finite volume method (MOUREH; FLICK, 2004). This method is widely studied and was also used by Moureh and Flick (2004) for the determination of the airflow and temperature pattern in a refrigerated truck loaded with pallets.

3.3.3.1 Airflow

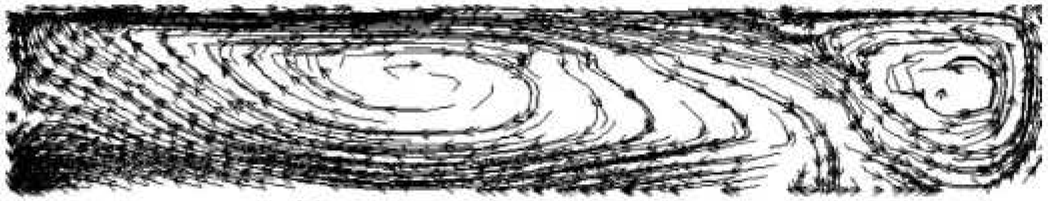
The airflow is important to the maintenance of the food characteristics and quality, so the study and validation of numerical simulations to characterize airflow behavior are important to help to control this parameter due to the complexity of the instrumentation needed in large enclosures (MOUREH; FLICK, 2005).

For a scale model, Moureh and Flick (2005) compared predicted values and experimental data of the airflow pattern inside an empty long slot-ventilated enclosure. Two different arrangements of air inlet were used, central and lateral position, and due to the common use of central air inlet, a small and a large section was studied for this case. The scale model had wooden walls, except one wall that was made of glass to allow measures of air velocity. The predicted values were obtained using FLUENT, and the governing equations were solved using the finite-volume method in a staggered grid system. Because of the high gradients near the inlet, outlet, and walls, it was used a high-density mesh in those areas. Most of the papers, studied by the authors, positioned the inlet facing the outlet, but it allows the creation of a strong pathway with a high and a low airflow velocity, in the front and rear section respectively. Concerning the turbulence modeling, the use of an inlet in the lateral reinforced the Coanda effect[1] and avoided the separation of the jet, allowing more homogeneous ventilation with better maintenance of temperature and concentration throughout the enclosure. The use of the second moment closure rather than the two-equation turbulence models to predict airflow patterns and flow separation was justified with experimental results that showed the complexity of the confined turbulent flow generated by the wall jet where adverse pressure gradients are present and the effect of anisotropy of turbulence fluctuations and streamline curvature are pronounced. The experimental compared with simulation, using tree different turbulence models, is shown in Figure 5.

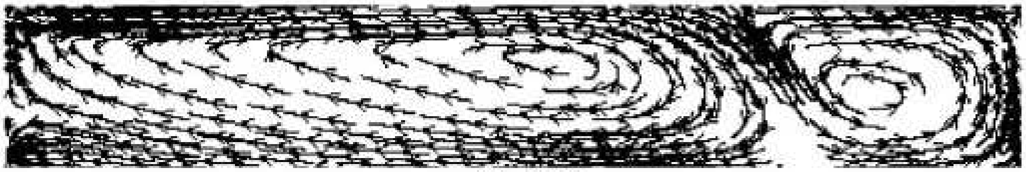
---

[1] Coanda effect is the phenomena in which a jet flow attaches itself to a surface and follows it, even if it curves away from the initial jet direction (SEO; OH; JANG, 2017).

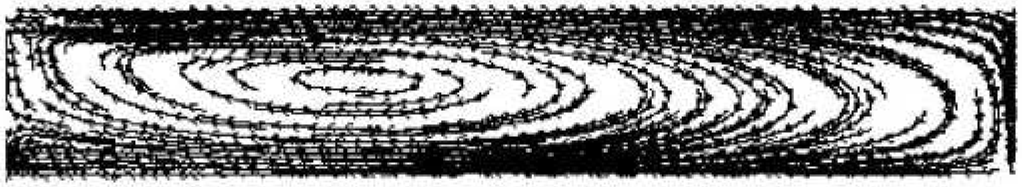
Figure 5 Airflow pattern on the symmetry plane.



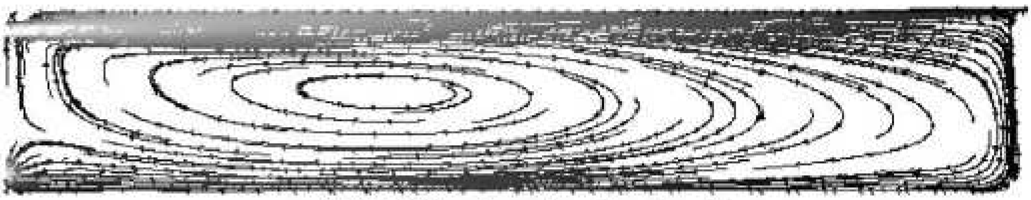
(a) Experiment (1080 measurement points)



(b) RSM



(c) High Reynolds k-ε model

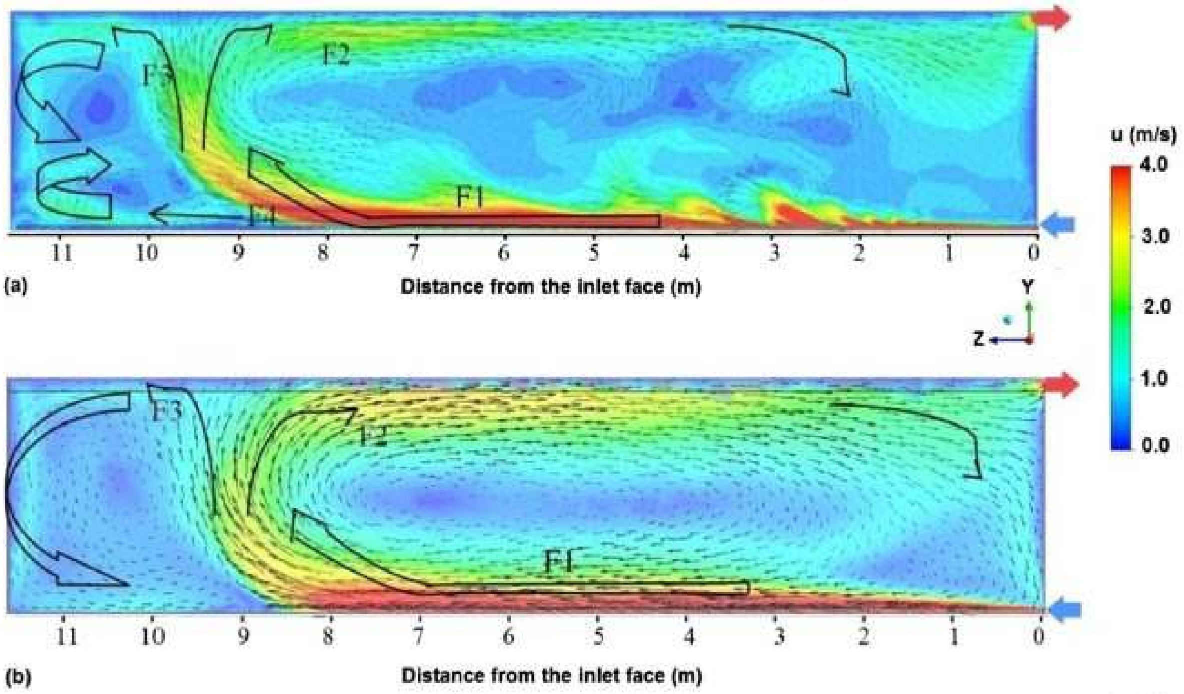


(d) LRN k-ε Lam-Bremhorst model

Source: Moureh and Flick (2005)

Getahun et al. (2018) simulated the airflow inside two different types of refrigerated shipping containers with different floor designs (T-bar and flat floor) and validated with experimental data. The assumptions made by the authors were: cooling unit was accounted by using the measured air velocity data at the inlet and outlet boundaries of the cooling unit; physical properties of the air were assumed constant; and outlet airflow velocity does not vary throughout the reefer unit. ANSYS DesignModeler Release 16.0 was used to develop the model geometry. For the T-bar floor design an extra finer mesh was used on narrow structures. ANSYS Meshing was used in the discretization of the domain and ANSYS Fluent Release 16.0 for the problem setup and simulation. The study showed the influence of the floor design on airflow distribution. The T-bar structure obtained a better result because increased proportion of vertical airflow when compared to flat floor design. The simulated airflow pattern is shown shown in Figure 6. The developed model can be applied to study other parameters, such as energy consumption and environmental conditions. However, more study needs to be made to improve the design with a packed refrigerated shipping containers.

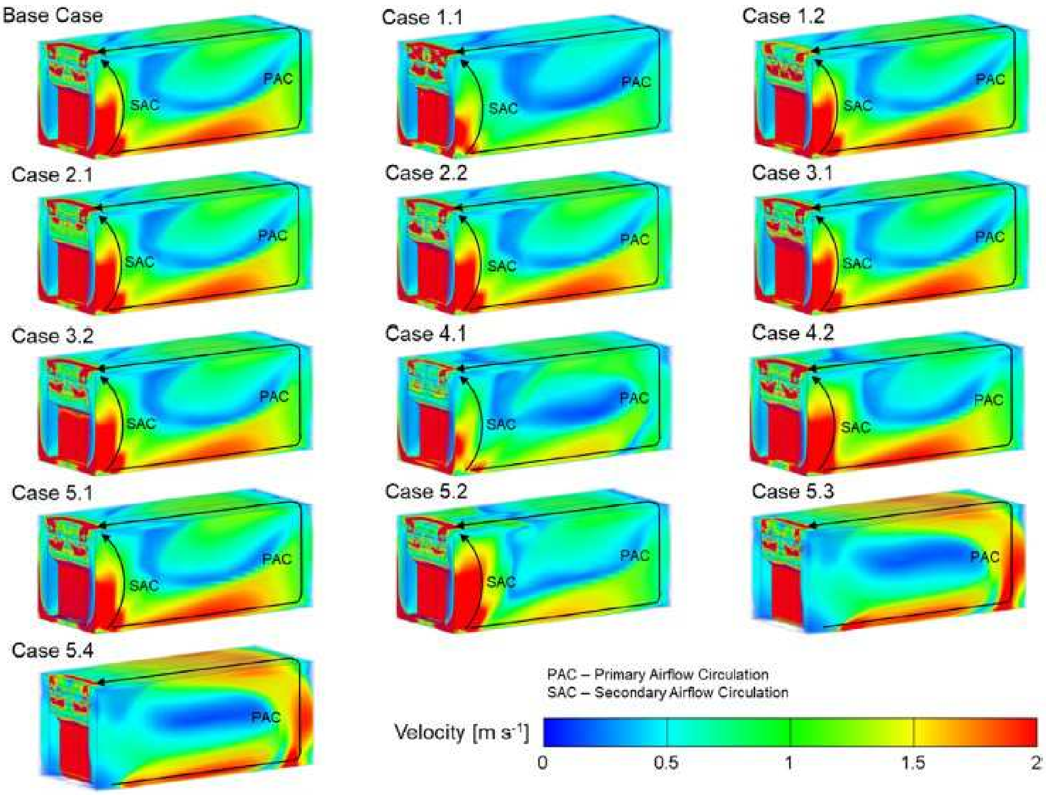
Figure 6 Airflow pattern on a section plane. (a) T-bar, (b) flat floor.



Source: adapted from Getahun et al. (2018)

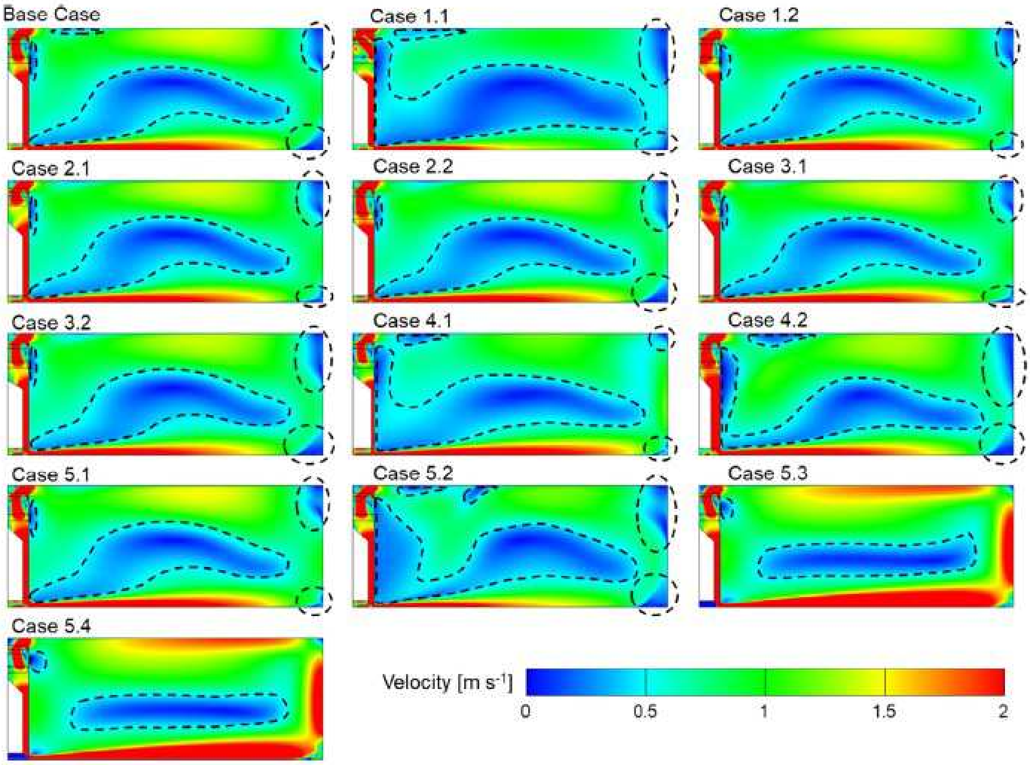
A study developed by Senguttuvan et al. (2020) reinforces the use of CFD to improve the airflow of a refrigeration container. Dimensions and design parameters were varied and fifteen different models of containers were considered. Governing equations describing the airflow are continuity and momentum equations. Those equations were numerically solved by the finite volume method with ANSYS FLUENT 19.1, and κ-ε turbulence model to approximate turbulence parameters. With the simulations, it was possible to see which refrigeration unit improved the airflow and also analyze the airflow pattern with the respective refrigeration unit. If this study were performed with experimental data, it would be much more difficult to obtain the results and it would be necessary more more time, money and experimental effort. Figure 7 and Figure 8 show the result of the simulation in twelve different cases made by the author. The cases 5.3 and 5.4 exhibited significantly hogher jet velocity, high vertical velocity distributions, improved Coanda effect, higher mass averaged velocity and reduced areas of dead zones in tne container. In cases 6.2 and 6.3 there was a significant improvement in the overall velocity characteristics, airflow distributions, and drastic reduction in dead zones inside the refrigerated containers. However, among all the cases, case 6.2 is the recommended, because has the highest mass averaged velocity in the container, 33,66% higher than the base case.

Figure 7 Comparison of volume-rendered velocity contour.



Source: Senguttuvan et al. (2020)

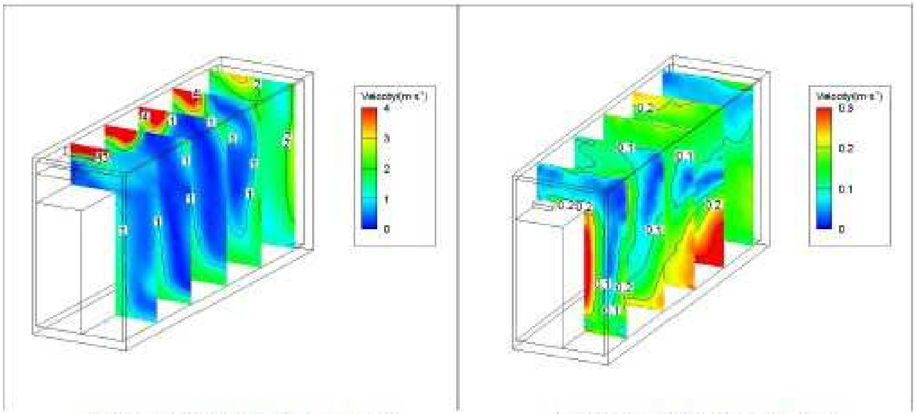
Figure 8 Comparison of velocity contour air symmetry plane.



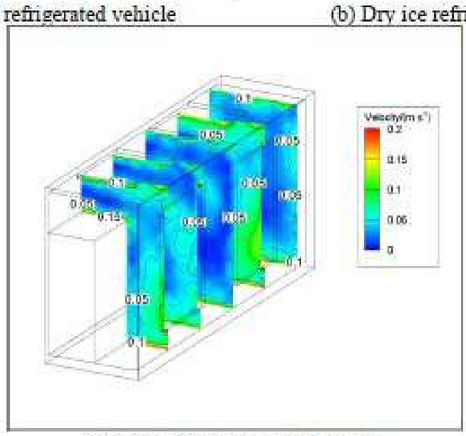
Source: Senguttuvan et al. (2020)

Different units of refrigeration were studied by Zhu and Xie (2020): mechanical, dry ice, and cold plate refrigeration. The authors performed a CFD analysis using FLUENT 19.0, to numerically simulate, using CAD to obtain the geometry of the truck, and Mesh in ANSYS 19.0 to generate the meshes. Number of meshes were set at 150000, 204000 and 247000. Different sizes of meshes were used for calculation, in order to prove that the simulation result was independent. Some assumptions were made to facilitate the analysis, such as the heat exchange rate between the external environment and the compartment of the refrigerated truck was even and constant, the air in the car was consistent with the boussinesq assumption, the parameters of the cold storage agent remain unchanged, the temperature distribution in the cold storage plate was uniform at the initial moment, was ignored the flow of the phase change cold storage medium in the PCM cold storage plate, and the initial temperature of frozen products in the compartment was -18ºC. For the mechanical refrigerated vehicle the κ-ε model was used, while the dry ice and the cold plate refrigerated vehicle refered to others' settings for numerical calculations of low-speed incompressible fluids in wich used κ-ε model, enhanced wall treatment, and full buoyancy effects. The postprocessing was made importing the results to Tecplot software. The authors concluded that the air temperature is related to the cold source temperature, but airflow influences the distribution of the temperature. A higher airflow produces a more uniform distribution of the temperature inside the truck, outstanding the importance of the study and optimization of the airflow. The result of the simulation in the tree different refrigeration system is illustrated in Figure 9.

Figure 9 Velocity fields of refrigerates vehicles



(a) Mechanical refrigerated vehicle (b) Dry ice refrigerated vehicle



(c) Cold plate refrigerated vehicle

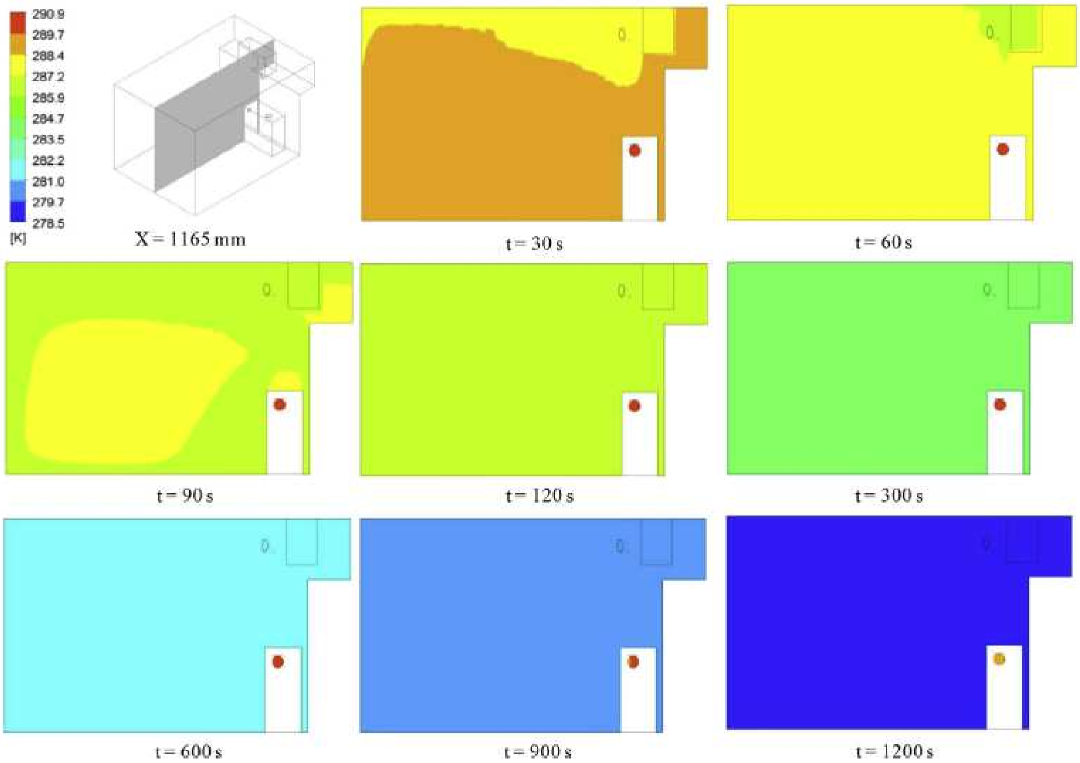
Source: Zhu and Xie (2020)

3.3.3.2 Temperature

Aiming to keep the quality and integrity of the food being transported, it is important to maintain the required product temperature range from production to consumption. However, keeping the ideal temperature still is a difficult task (NOVAES et al., 2015). Thus, CFD validated models can be used to study the efficiency of the system used and to help improving it.

In Umeno et al. (2015) research paper it was developed a CFD model to simulate temperature distribution inside a refrigerated container to store fresh product not loaded and validated with experimental results. The containers were made of stainless steel and was equipped with nanomist humidifier and ultrasonic humidifier. ANSYS FLUENT 14.0 (ANSYS Inc.) was used to numerically solve the governing mass, momentum and energy equations. The author used empirical equations, written in Microsoft visual studio express 2012 and embedded into Fluent solver code as an user-defined function during the simulation, in order to support the CFD software to simulate temperature distribution. Pressure jump at the outlet of refrigerator was computed using trial error method by means of adjusting the pressure parameter in FLUENT until the predicted air velocities fitted well with experimental data. The simulation was made in an unsteady state and the total CPU time of calculation was 38 and 12.3 hours for nanomist and ultrasonic, respectively. Fo all equations, except for the energy one, first-order implicit time stepping was used with step size of 1s and thirty iterations were performed for each step time to achieve normalized residues below 10^{-3}. Hexahedral and tetrahedral meshes with a total number of 472346 and 195886 of mesh elements were used for nanomist and ultrasonic chamber. The results were evaluated by the temperature distribution and monitored over time, as shown in Figure 10, had an error of 0.66K for nanomist and 0.95K for ultrasonic when compared to experimental results. The simulation was successfully validated and can help to predict temperature distribution in refrigerated containers similar to the studied ones.

Figure 10 Changes in predicted temperature profile over time on YZ surface in nanomist container.



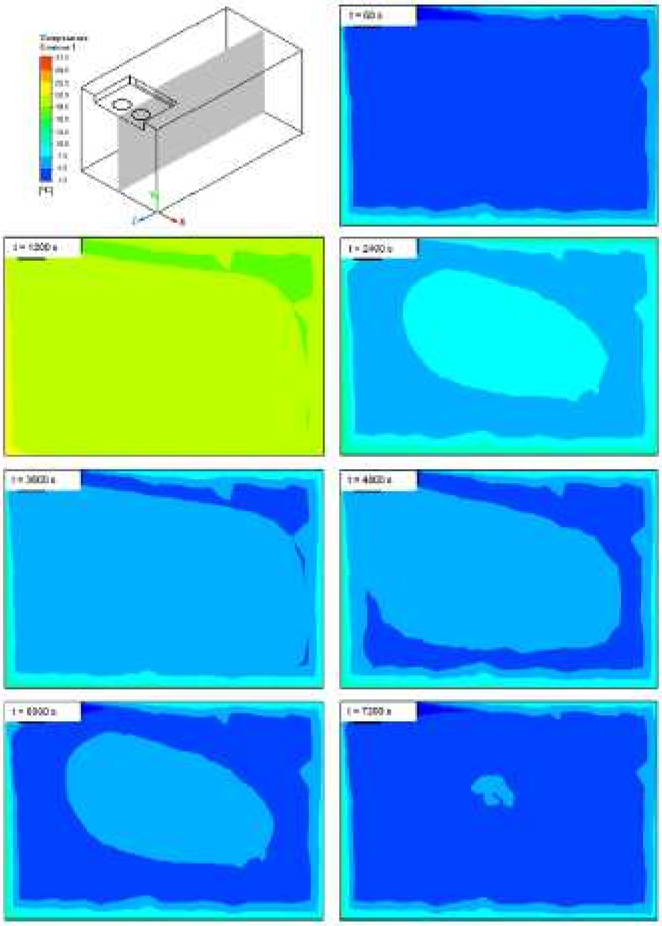
t= 30 s
t= 60 s
t= 90 s
t= 120 s
t= 300 s
t= 600 s
t= 900 s
t= 1200 s

Source: Umeno et al. (2015)

Design characteristics of the container influence the refrigeration inside. Therefore, Kayansayan, Alptekin, and Ezan (2017) numerically studied conjugated heat transfer in a ceiling-slot refrigerated container with heat-conductive walls to analyze temperature distribution and determine ventilation characteristics. The parameters used to evaluate temperature distribution effectiveness were the container shape factor, the inlet air slot width, and the Reynolds number of supplied cold air flow. The convective heat transfer inside the container coupled to heat conduction through opaque walls was analyzed, and six configurations were studied. The finite volume technique was used to discretize the governing equations describing the flow and heat transfer. The grid node positions were calculated using a stretching function in which nodes close to the walls are denser than the meshes at the central region of the container, and a Grid Convergence Index (GCI) was used to the grid refinement effect. The experimental data obtained in the study of Moureh and Flick (2005) was used by the authors to compare the results. Turbulent flow conditions were used to evaluate the heat gain through the container conductive walls. An optimum velocity of injection that leads to a maximum effectiveness was noted. The correlations for the mean Nusselt and Reynolds numbers represented the numerical results with a maximum of 14.54% of error. According to the authors, the CFD analysis allows a better understanding and prediction of heat loads and thermal characteristics of airflow throughout transportation.

Using the CFX module of ANSYS WORKBENCH 16.0, Jara et al. (2019) estimated the temperature distribution inside an empty refrigerated vehicle for food transportation using a CFD Shear Stress Transport calculation model. The authors used sandwich panels insulation made of galvanized sheet in the outer, a food-grade stainless sheet in the inner, and expanded polyurethane in between them. To the simulation, triangular elements were used in the meshes, and, as the temperature inside the vehicle wasn't constant, an expression of temperature versus time was entered in the software, and the convective coefficient was calculated through the walls. The largest absolute error given by the simulation was in the first 540 seconds after the precooling initiation (3.11ºC), but when the desired temperature was reached, the maximum variation was 0.71ºC. The greatest variations were observed on the floor and at the exit, where the temperature does not remain constant, due to factors as the engine and exhaust pipe. Overall, the results obtained in the simulation were acceptable. The study of temperature homogeneity is important to the conservation of the food inside the truck therefore, this study allows to use the technique as a primary analysis instrument and the optimization of the minimum stabilization time of the temperature. In Figure 11 exhibits the temperature simulation on a point of the container along the time.

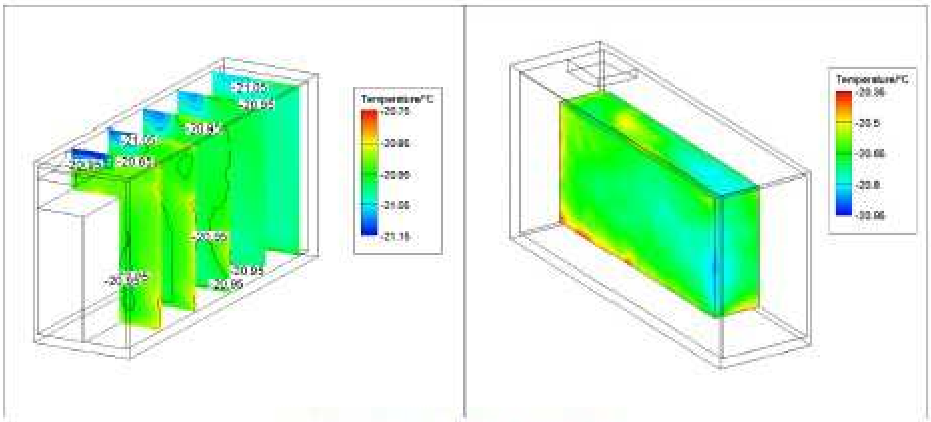
Figure 11 Temperature profiles on the plane X=-0.8m with times of t = 60s, t = 1200s, t = 2400s, t = 3600s, t = 4800s, t = 6000s, t = 7200s



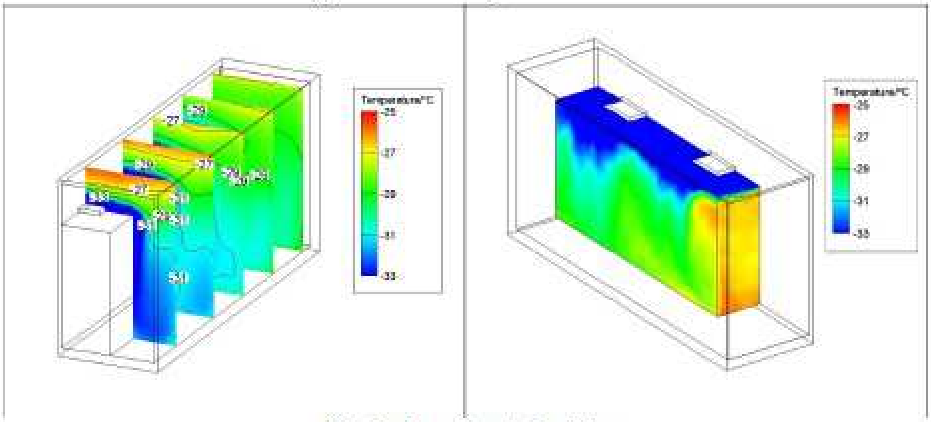
Source: Jara et al. (2019)

On Zhu and Xie (2020) study, they also analyzed the temperature field with the three different mechanical systems. The results showed that temperature was more uniform with traditional mechanical refrigerated system, but the airflow in this case was heterogeneous in this case. Location of cold source has an influence on the air temperature distribution, and it was found a positive relationship between the could source temperature and the air. The simulation results are shown in Figure 12 and the conclusions were made observing the airflow and temperature field. Even with a heterogeneity of the airflow the temperature in the compartment of traditional mechanical refrigerated vehicle is the most uniform. In the dry ice refrigerated car the airflow was uniform, but the temperature had a large difference along the vehicle. The airflow velocity inside the plate refrigerator is the lowest and the temperature is uniform, but more cold plates are needed to get enough heat transfer area due to the low airflow in wich reduces the area of goods on the vehicle.

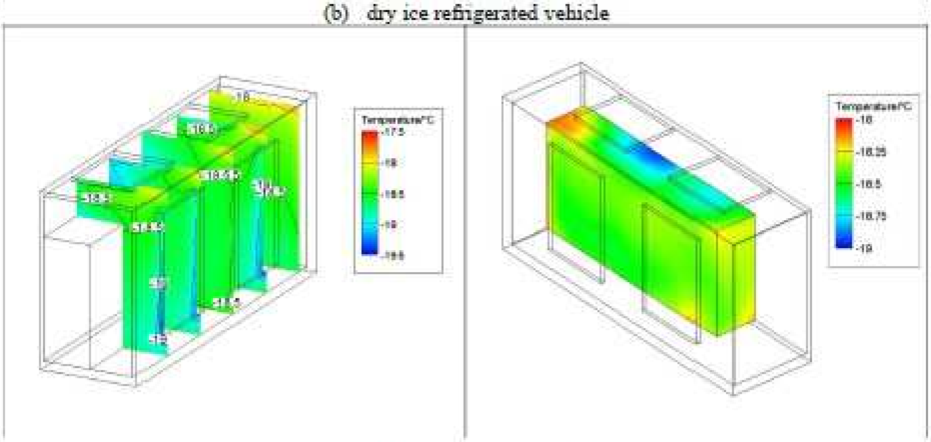
Figure 12 Temperature fields of refrigerated vehicle and cargo surface.



(a) mechanical refrigerated vehicle



(b) dry ice refrigerated vehicle



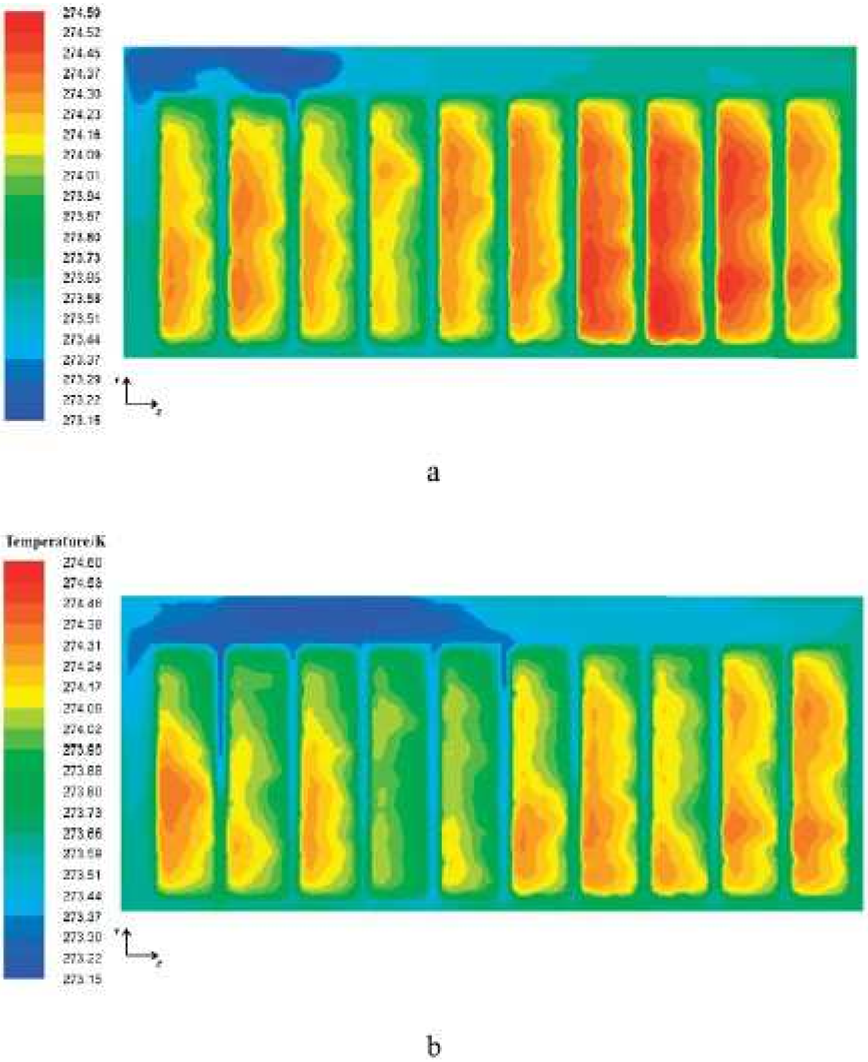
(c) cold plate refrigerated vehicle

Source: Zhu and Xie (2020)

3.3.3.3 Loaded truck

Han et al. (2016) utilized a validated 3D CFD model of a refrigerated compartment to predict the temperature variation in a cargo during refrigeration phase and transportation and compare with experimental results. The refrigeration truck used was for short distance transportation, its insulation was made of polyurethane foam, the cooling unit was situated in the front of the vehicle and it was loaded with two rows of 20 groups of goods. To verify the influence of the air duct two configurations were investigated, one without an air duct and another with air duct and two different inlet locations. For the mathematical model stress transport (SST) κ-ω model was solved using FLUENT (ANSYS, Inc. 2009). To simplify calculation some assumptions were made: the air is considered an incompressible fluid with constants properties, the air is regarded as a Newtonian fluid and a Boussinesq fluid, the stack goods is considered to be an isotropic porous medium, the mass transfer of the moisture in the air to the product is neglected and heat transfer in the stack of goods is governed by conduction and convection only. The use of the air duct led to a significant improvement of the airflow, resulting in a more uniform temperature distribution and it can be analyzed by comparing the simulations in Figure 13. A 0ºC temperature satisfies the requirement of the cargo and consume less energy, thereby, this work provided reliable theoretical arguments to improve the temperature distribution and reduce unnecessary energy consumption.

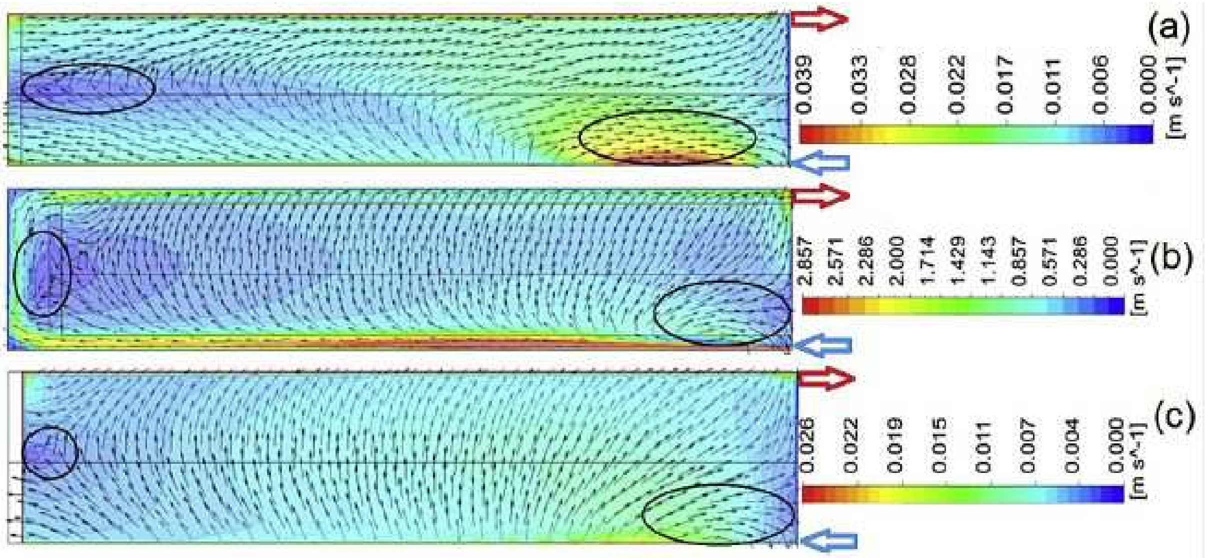
Figure 13 Temperature distribution (a) without air duct and (b) with air duct.



Source: Han et al. (2016)

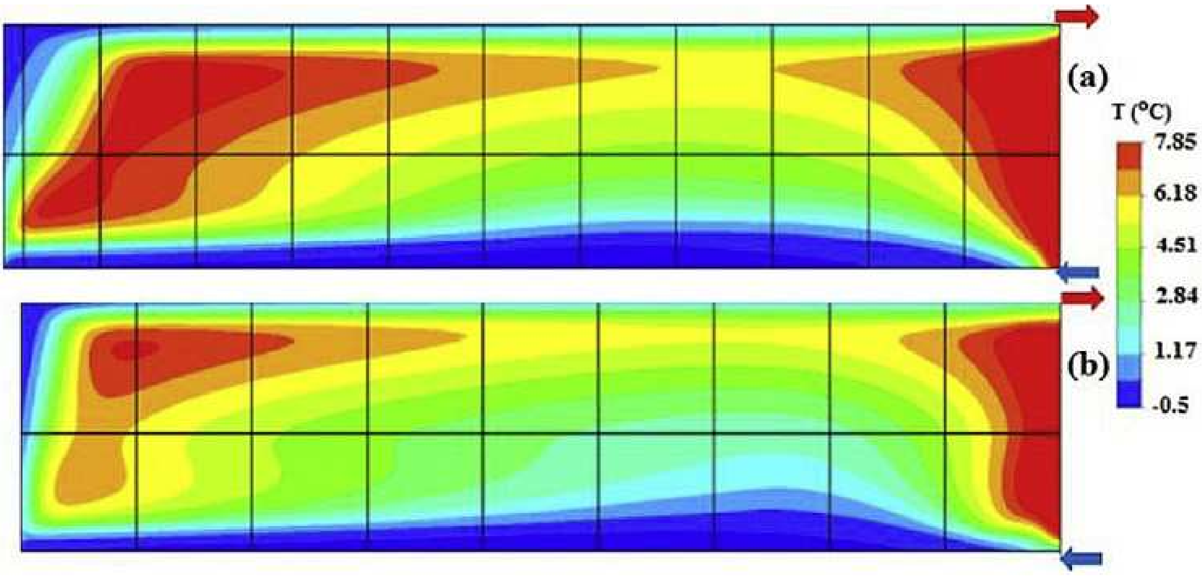
Getahun et al. (2017) studied a ship container loaded with fruit packed refrigerated, developing and validating a CFD model to predict airflow and heat transfer inside it. The insulation was made of polyurethane sandwiched between two metal sheets, cold air inlet situated in the bottom and outlet in the top of the refrigerated unit. Inlet curve was used to guide the airflow between the T-bars, and the pallets were arranged in two rows with a total of 15 tons of fruit. To develop a zonal porous medium CFD model, including T-bar flow and resistance to airflow of wooden pallets, the authors used: volume averaging technique to develop a single-phase porous medium model of airflow and heat transfer; κ-ω SST model to calculate turbulence; SIMPLE discretization scheme was used for pressure-velocity coupling; Second-Order Upwind discretization for momentum, specific dissipation rate, and energy calculation. The authors used ANSYS Design-Modeler for the geometry, ANSYS Meshing Release 16.0 for discretization of the domain, and ANSYS® Fluent ™ Release 16.0 to the problem set up and simulation. In air inlet and outlet, T-bar floor structures and regions with large velocity gradients fine meshes were applied. Wind tunnel experiments to characterize airflow resistance of the pallets with fruit and acquire pressure drop *vs.* velocity data were used to develop the model. The simulation was able to reproduce airflow and temperature profiles showing high and low airflow and cooling regions, as can be seen in Figure 14 and Figure 15. It also showed a cooling heterogeneity, that could be caused by the absence of vent holes on the bottom side of the packaging, and high airflow. A fully loaded container has a high complexity system, and because of that, the comparison of the model with data from a full-scale container packed with apple was considered reasonable. However, a more realistic and integrated assessment taking various internal and external factors that incorporate fruit quality parameters into account could improve the model.

Figure 14 Airflow profile inside a fully loaded reregerated container. Colour contours show magnitude of air velocity and arrows show the local airflow direction on vertical plane bisecting pallets in row 1 (a), the air gap between the two rows (b) and pallets in row 2 (c).



Source: Getahun et al. (2017)

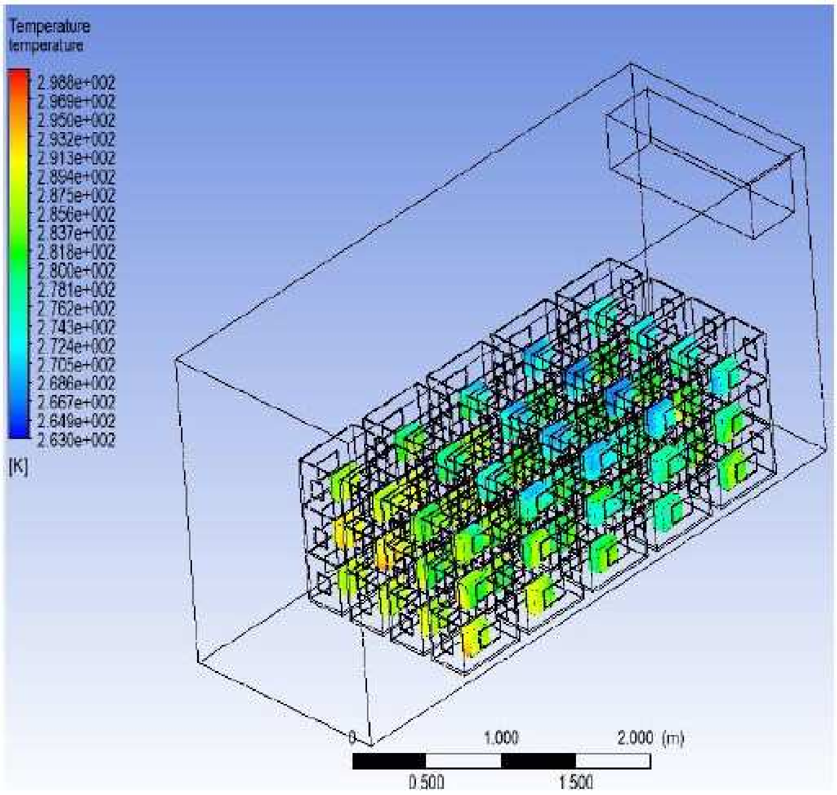
Figure 15 Simulated profile of produced temperature at 24h on vertical plane (YZ-plane) bisecting row 1 (a) and row 2 (b) inside a refrigerated container.



Source: Getahun et al. (2017)

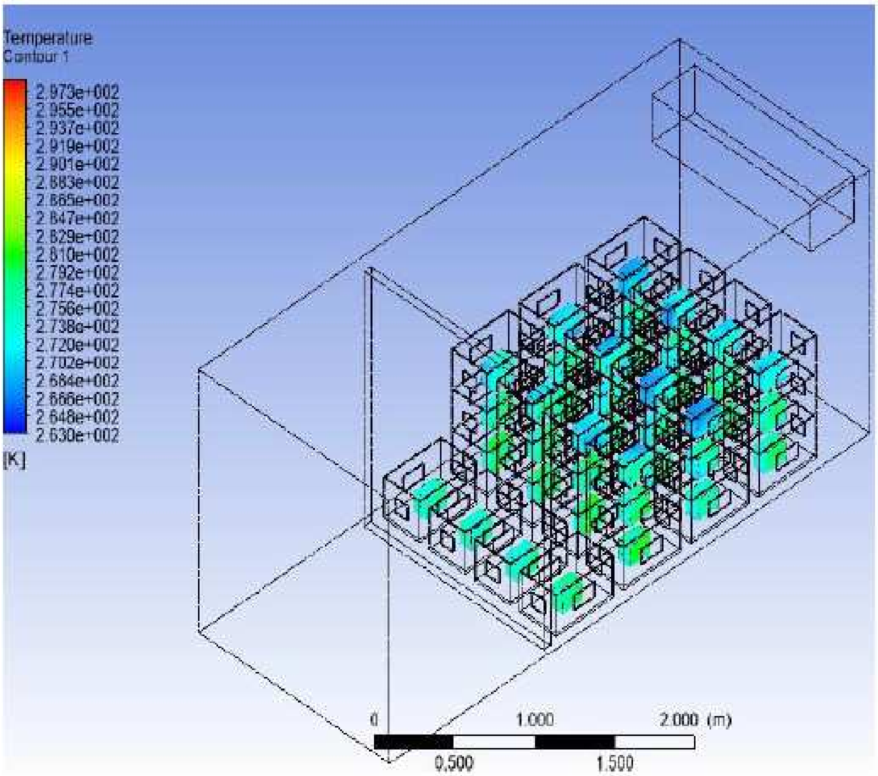
Imtiaz, Bhuiyan and Rahman (2020) researched different partitions arrangements with horizontal and vertical shelves, inside a refrigerated vehicle with experimental and CFD simulations observing a better air circulation and more uniform temperature distribution when a vertical partition was used. Numerical analysis was made using ANSYS Fluent, shear stress transport and SST-k-omega model was used for turbulence modeling. Effects of radiation from outside was ignored and the property of aluminum was used for the partitions. The results showed an improvement in the modified designs than the conventional ones. Vertical portioning resulted in a better thermal performance and significantly increase of the percentage of products stored within the acceptable temperature, as can be seen in Figure 16 compared with Figure 17. Figure 18 shows streamlines in two different arrangements and as can be seen in simulation (b), the the vertical partition led to a more uniform air circulation.

Figure 16 Temperature distribution in a typical compartment design.



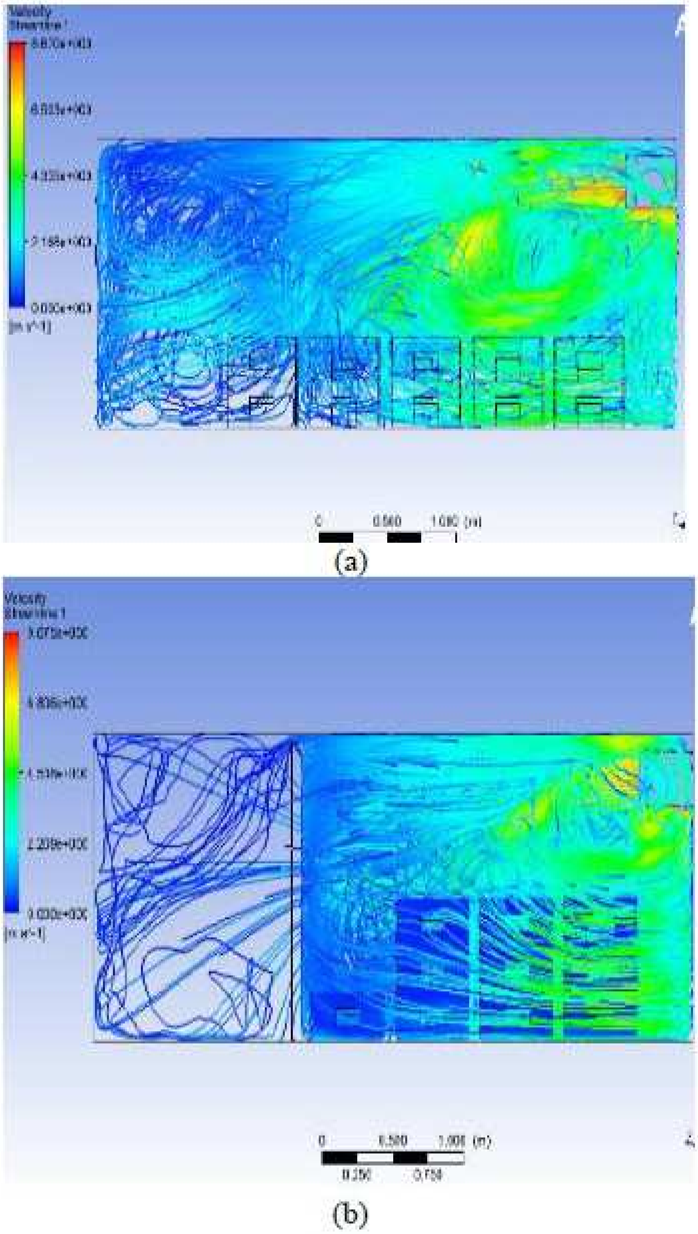
Source: Imtiaz, Bhuiyan and Rahman, (2020)

Figure 17 Temperature distribution for the best vertical partitioning design.



Source: Imtiaz, Bhuiyan and Rahman, (2020)

Figure 18 Streamlines for the (a) traditional storage design and (b) modified chamber design with vertical partition.



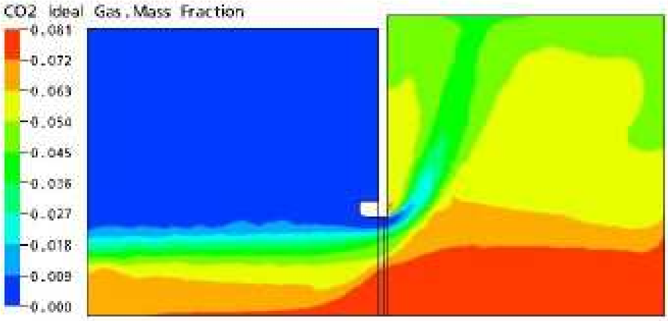
Source: Imtiaz, Bhuiyan and Rahman, (2020)

3.3.3.4 Air infiltration

A truck can deliver cargo in more than one place, which implies constant openings of the door along the route. The opening of the door leads to a heat gain causing a heterogeneity of the airflow and temperature inside the body, which can result in significant energy consumption. The door opened even for a short period can result in an air infiltration representing more than half of the refrigeration load (RAI; SUN; TASSOU, 2019; GONÇALVES; COSTA; LOPES, 2019; NOVAES et al., 2015).

Gonçalves, Costa, and Lopes (2019) studied the air movement through the doorway of a refrigerated room creating a model and validating it with an experimental approach. For the simulation the authors used ANSYS-CFX code and the κ-ω SST turbulence. Empirical models tend to overpredict the air infiltration, which can affect the view of the problem. The results showed that the numerical model used can predict reasonably well the air infiltration, and was also confirmed with the use of the Gosney and Olama model to estimate the heat gain due to air exchange. The higher prediction error was 12%, corresponding to the largest door height. In Figure 19 is possible to observe the increase concentration of CO_2 after the door is opened.

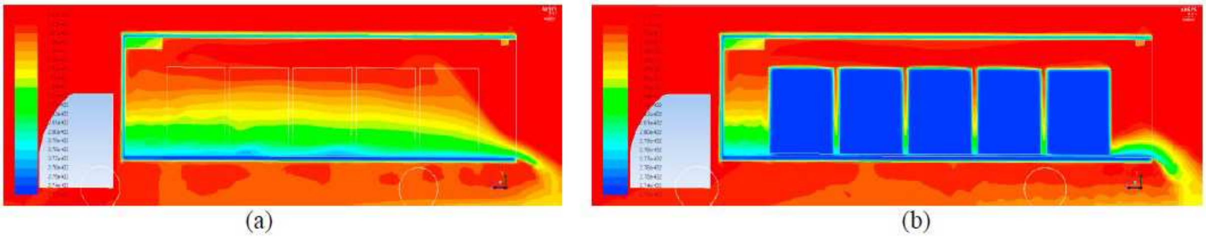
Figure 19 Simulation of CO_2 concentration field in the symmetry plane at t=60s after the door was opened.



Source: Gonçalves, Costa, and Lopes (2019)

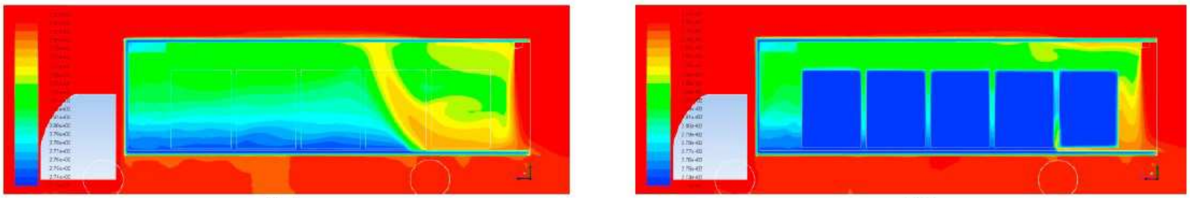
An alternative to reduce the impact of air infiltration is the use of an air curtain. A study using the CFD approach showed that it can help reducing energy consumption by almost 48%. The simulation was made using ANSYS ICEM CFD 14.5 to create the mesh, ANSYS Fluent 14.5 to implement the mesh domain and simulate, and solutions were obtained using Reynolds-averaged equation for conservation of mass, momentum, and energy, and κ-ε model for turbulence effects (RAI; SUN; TASSOU, 2019a). Figure 20 and Figure 21 show the temperature profile, while Figure 22 and Figure 23 illustrate the airflow distribution, inside the truck without and with air curtain, respectively.

Figure 20 Temperature distribution inside a refrigerated truck with the door opened: (a) mid-plane view, (b) product mid-plane view.


(a) (b)

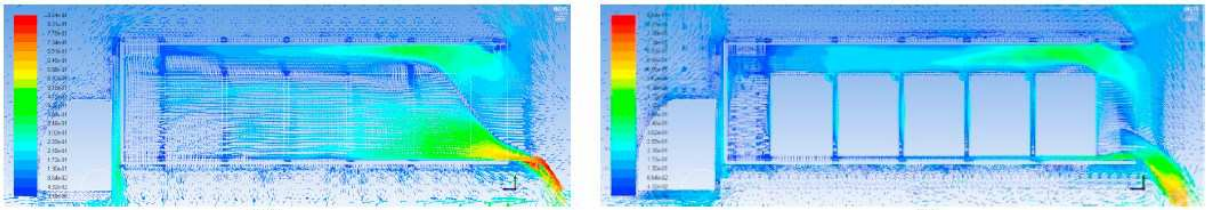
Source: Rai, Sun and Tassou (2019a)

Figure 21 Temperature distribution inside a refrigerated truck with the door opened and with air curtain: (a) mid-plane view, (b) product mid-plane view.


(a) (b)

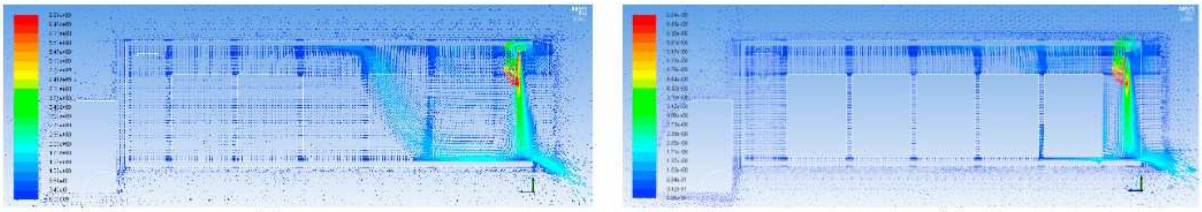
Source: Rai, Sun and Tassou (2019a)

Figure 22 Velocity changes inside a refrigerated truck with the door opened: (a) mid-plane view, (b) product mid-plane view.


(a) (b)

Source: Rai, Sun and Tassou (2019a)

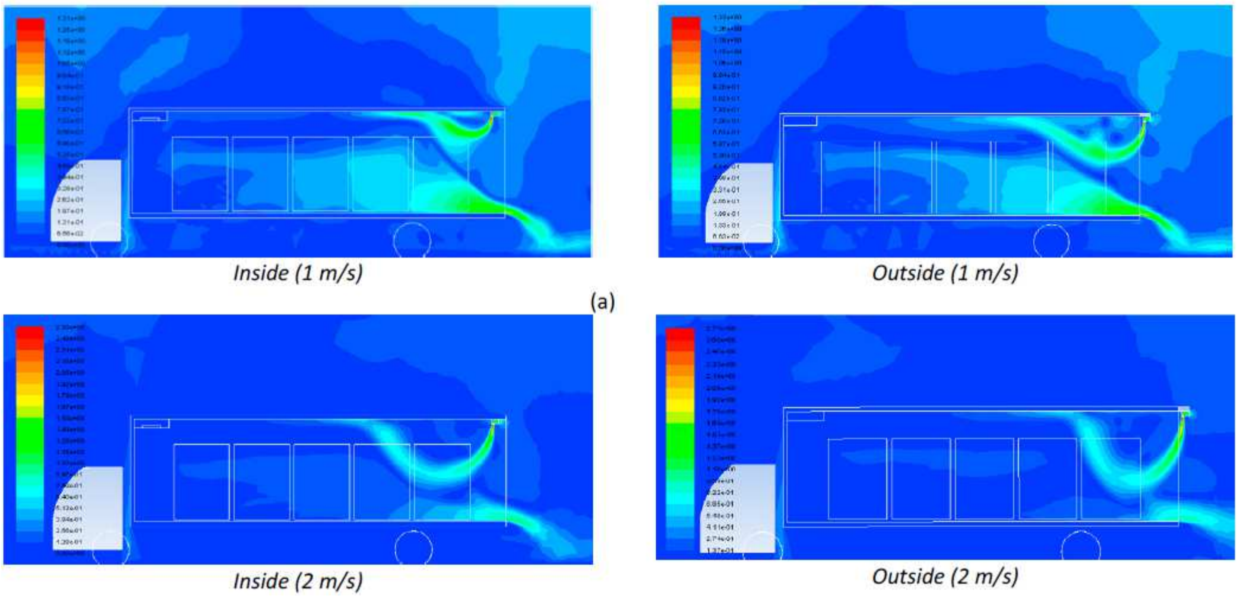
Figure 23 Velocity changes inside a refrigerated truck with the door open and with air curtain: (a) mid-plane view, (b) product mid-plane view.


(a) (b)

Source: Rai, Sun and Tassou (2019a)

Rai, Sun, and Tassou (2019b) studied the use of an air curtain in two different positions, inside and outside the chamber, aiming to improve its performance. The thermal insulation used was polyurethane. The model was created using CAD software Solidworks 2017, imported in Ansys ICEM CFD 14.5 to generate meshes (hexagonal mesh in the truck and tetragonal mesh in the atmosphere), and simulation in Ansys Fluent 14.5. The authors used the Reynolds-averaged equation of conservation of mass, momentum, and energy, and the k-ε model, to estimate a solution for the problem and the turbulence effect, respectively. The air curtain placed outside had a better energy performance and due to the Coanda effect, the air bends at the door opening and reaches the floor, as showed in Figure 24.

Figure 24 Velocity contours of air curtain placed inside and outside at discharge velocity (a) 1m/s and (b) 2m/s



Source: Rai, Sun, and Tassou (2019b)

4 CONCLUSION

The improvement of the refrigeration truck means to reduce losses and keep the quality of the product. For this, the refrigeration vehicle must have the right specifications to maintain the optimal temperature all over the cargo. Therefore, there are different refrigeration systems, insulations and inside design that can be used. Among these, the most used refrigeration system and insulation is the mechanical system and polyurethane, respectively. To make a CFD simulation is important to take into account all these characteristics to select domain, boundary and initial conditions while using the preprocessor, solver and postprocessor.

Furthermore, studies of refrigeration trucks CFD simulations showed that it can be used for different purposes, such as improve the refrigeration unit, airflow, temperature homogeneity, insulation, arrangement of the cargo, inlet and outlet position or the use and positioning of air curtain. To approximate turbulence parameters SST κ-ω turbulence model is the most used, but κ-ω and κ-ε models are also used depending on the characteristics of the problem. There are a lot of equations, parameters and conditions that have to be analyzed when performing CFD modeling. Therefore, usually a first study of the empty truck is realized prior to a more complex like a loaded truck. In order to simplify the solution some assumptions can be made, such as air be considered an inconpressible Newtonian fluid,and Boussinesq approximation, neglect irradiation and porous medium to approximate the cargo. Coanda effect have a big influence in the airflow, so it is important to take it into account the position of the air inlet and outlet and analyze the results. Each truck has its own characteristics, but once the model is validated it can be used to reduce time and cost to improve a refrigerated truck. Recent studies have been made on the topic with more complex systems, but there is still space for more improvements.

The development of the computational power enables to make even more detailed simulations, so future researches can consider even more characteristics of a loaded truck considering the outside temperature, time and distance to travel because there is no much studies specifically related to trucks. Also, further studies can consider different configurations of inlet and outlet, cargo configuration, air curtain, a variety of insulations, more external factors and different refrigeration units.

REFERENCES

ARTUSO, P. et al. Dynamic modeling and thermal performance analysis of a refrigerated truck body during operation. **International Journal of Refrigeration**, v. 99, p. 288–299, 2019. Disponível em: <https://doi.org/10.1016/j.ijrefrig.2018.12.014>.

FLUENT ANSYS. Ansys Fluent Theory Guide. **ANSYS Inc., USA**, v. 15317, n. November, p. 724–746, 2013.

BADIA-MELIS, R.et al. I. **New trends in cold chain monitoring applications - A review**. Food Control, v. 86, p. 170–182, 2018. Disponível em: <https://doi.org/10.1016/j.foodcont.2017.11.022>.

BAPTISTA, P. **Higiene e Segurança Alimentar no Transporte de Produtos Alimentares**. [s.l.] Guimarães Projecto Gráfico e Design Forvisão, S.A., 2006.

BERNSTEIN, M. C.. **Comparação entre Equipamento Tradicional e Proposta de Sistema Otimizado para Refrigeração De Bebidas**. 2017. Universidade Tecnológica Federal do Paraná, *[S. l.]*, 2017.

BRASIL. **Rodovias Federais**. Ministério da Infraestrutura, 2019. Disponível em <https://antigo.infraestrutura.gov.br/rodovias-brasileiras.html>. Acesso em: 23 de junho de 2021

CHAN, K. C. et al. Experiment verified simulation study of the operating sequences on the performance of adsorption cooling system. **Building Simulation**, v. 8, n. 3, p. 255–269, 2015.

DALKILIÇ, A. S. et al. **Effect of refrigerant type and insulation thickness on refrigeration systems of land and sea vehicles**. Strojniski Vestnik/Journal of Mechanical Engineering, v. 62, n. 4, p. 252–259, 2016.

FAO. 30% DE TODA A COMIDA PRODUZIDA NO MUNDO VAI PARAR NO LIXO. Nações Unidas Brasil. Disponível em <https://nacoesunidas.org/fao-30-de-toda-a-comida-produzida-no-mundo-vai-parar-no-lixo/>. Acesso em: 15 de setembro de 2019.

FIORETTI, R.; PRINCIPI, P.; COPERTARO, B.. A refrigerated container envelope with a PCM (Phase Change Material) layer: Experimental and theoretical investigation in a representative town in Central Italy. **Energy Conversion and Management**, *[S. l.]*, v. 122, p. 131–141, 2016. DOI: 10.1016/j.enconman.2016.05.071. Disponível em: http://dx.doi.org/10.1016/j.enconman.2016.05.071.

GAO, P. et al. **Optimization and performance experiments of a MnCl2/CaCl2–NH3 two-stage solid sorption freezing system for a refrigerated truck**. International Journal of Refrigeration, v. 71, p. 94–107, 2016.

GENG et al. Review of experimental research on Joule–Thomson cryogenic refrigeration system. **Applied Thermal Engineering**, *[S. l.]*, v. 157, n. April, p. 113640, 2019. DOI: 10.1016/j.applthermaleng.2019.04.050. Disponível em: <https://doi.org/10.1016/j.applthermaleng.2019.04.050>.

GETAHUN, S. et al. Experimental and numerical investigation of airflow inside refrigerated shipping containers. **Food and Bioprocess Tecnology**, p. 1164–1176, 2018. Disponível em: <https://doi.org/10.1007/s11947-018-2086-5>.

GETAHUN, S. et al.. **Analysis of airflow and heat transfer inside fruit packed refrigerated shipping container: Part I – Model development and validation**. Journal of Food Engineering, v. 203, p. 58–68, 2017. Disponível em: <http://dx.doi.org/10.1016/j.jfoodeng.2017.02.010>.

GONÇALVES, J. C.; COSTA, J. J.; LOPES, A. M. G. Analysis of the air infiltration through the doorway of a refrigerated room using different approaches. **Applied Thermal Engineering**, v. 159, p. 113927, 2019. Disponível em: <https://doi.org/10.1016/j.applthermaleng.2019.113927>.

HAN, J. W. et al. Computational Fluid Dynamics Simulation to Determine Combined Mode to Conserve Energy in Refrigerated Vehicles. **Journal of Food Process Engineering**, v. 39, n. 2, p. 186–195, 2016.

HAN, J. W. et al. **CFD Simulation of Airflow and Heat Transfer During Forced-Air Precooling of Apples**. Journal of Food Process Engineering, v. 40, n. 2, p. 1–11, 2017.

IMTIAZ, F.; BHUIYAN, M. S. A.; RAHMAN, M. A. Assessment of Thermal performance of Transport Refrigeration Vehicles by Modifications in the Compartment Design and Storage Pattern. **Proceedings of International Exchange and Innovation Conference on Engineering & Sciences (IEICES)**, v. 6, n. October, p. 284–291, 2020.

JAMES, C. Food transportation and refrigeration technologies-design and optimization. In: **Sustainable Food Supply Chains: Planning, Design, and Control through Interdisciplinary Methodologies**. [s.l.] Elsevier Inc., 2019. p. 185–199.

JARA, P. B. T. et al. Thermal behavior of a refrigerated vehicle: Process simulation. **International Journal of Refrigeration**, v. 100, p. 124–130, 2019. Disponível em: <https://doi.org/10.1016/j.ijrefrig.2018.12.013>.

KAYANSAYAN, N.; ALPTEKIN, E.; EZAN, M. A. Analyse thermique du débit d'air à l'intérieur d'un conteneur réfrigéré. **International Journal of Refrigeration**, v. 84, p. 76–91, 2017. Disponível em: <https://doi.org/10.1016/j.ijrefrig.2017.08.008>.

LAKATOS, Á.; KOVÁCS, Z. Comparison of thermal insulation performance of vacuum insulation panels with EPS protection layers measured with different methods. **Energy and Buildings**, *[S. l.]*, v. 236, 2021. DOI: 10.1016/j.enbuild.2021.110771.

LI, Y. et al. Review on research and application of phase change materials in cold storage refrigerator. **IOP Conference Series: Earth and Environmental Science**, v. 766, n. 1, p. 012094, 2021.

LIU, M.; SAMAN, W.; BRUNO, F. Development of a novel refrigeration system for refrigerated trucks incorporating phase change material. **Applied Energy**, *[S. l.]*, v. 92, p. 336–342, 2012. DOI: 10.1016/j.apenergy.2011.10.015. Disponível em: <http://dx.doi.org/10.1016/j.apenergy.2011.10.015>.

LONG, J. et al. Review of researches on coupled system and CFD codes. **Nuclear Engineering and Technology**, n. xxxx, 2021. Disponível em: <https://doi.org/10.1016/j.net.2021.03.027>.

MAIORINO, A. et al. The thermal performances of a refrigerator incorporating a phase change material. **International Journal of Refrigeration**, *[S. l.]*, v. 100, p. 255–264, 2019. DOI: 10.1016/j.ijrefrig.2019.02.005. Disponível em: https://doi.org/10.1016/j.ijrefrig.2019.02.005.

MEJJAOULI, S.; BABICEANU, R. F. RFID-wireless sensor networks integration: **Decision models and optimization of logistics systems operations**. Journal of Manufacturing

Systems, v. 35, p. 234–245, 2015. Disponível em: <http://dx.doi.org/10.1016/j.jmsy.2015.02.005>.

MOUREH, J.; FLICK, D. Airflow characteristics within a slot-ventilated enclosure. **International Journal of Heat and Fluid Flow**, *[S. l.]*, v. 26, n. 1, p. 12–24, 2005. DOI: 10.1016/j.ijheatfluidflow.2004.05.018.

MOUREH, J.; FLICK, D. **Airflow pattern and temperature distribution in a typical refrigerated truck configuration loaded with pallets**. International Journal of Refrigeration, v. 27, n. 5, p. 464–474, 2004.

MOUSAZADE, A.; RAFEE, R.; VALIPOUR, M. S. Thermal performance of cold panels with phase change materials in a refrigerated truck. **International Journal of Refrigeration**, *[S. l.]*, v. 120, p. 119–126, 2020. DOI: 10.1016/j.ijrefrig.2020.09.003. Disponível em: https://doi.org/10.1016/j.ijrefrig.2020.09.003.

NORTON, T.; SUN, D. W. **Computational fluid dynamics (CFD) - an effective and efficient design and analysis tool for the food industry: A review**. Trends in Food Science and Technology, v. 17, n. 11, p. 600–620, 2006.

NORTON, T.; SUN, D.-W. An Overview of CFD Applications in the Food Industry. In: SUN, D.-W. (Ed.). **Computational Fluid Dynamics in Food Processing**. [s.l.] Taylor & Francis Group, LLC, 2007. p. 2–36.

NOVAES, A. G. N. et al. Thermal performance of refrigerated vehicles in the distribution of perishable food. **Pesquisa Operacional**, v. 35, n. 2, p. 251–284, 2015.

PAN, Q.; PENG, J.; WANG, R. Application analysis of adsorption refrigeration system for solar and data center waste heat utilization. **Energy Conversion and Management**, *[S. l.]*, v. 228, n. xxxx, p. 113564, 2021. DOI: 10.1016/j.enconman.2020.113564. Disponível em: <https://doi.org/10.1016/j.enconman.2020.113564>.

RAI, A.; SUN, J.; TASSOU, S. A. Numerical investigation of the protective mechanisms of air curtain in a refrigerated truck during door openings. **Energy Procedia**, v. 161, n. 2018, p. 216–223, 2019a. Disponível em: <https://doi.org/10.1016/j.egypro.2019.02.084>.

RAI, A.; SUN, J.; TASSOU, S. A. Three-dimensional investigation on the positioning of air curtain on its effectiveness in refrigerated vehicles used for food distribution. **Energy Procedia**, v. 161, p. 224–231, 2019b. Disponível em: <https://doi.org/10.1016/j.egypro.2019.02.085>.

RELATÓRIO DOS LEVANTAMENTOS FUNCIONAIS DAS RODOVIAS FEDERAIS. DNIT. Disponível em <http://www.dnit.gov.br/download/planejamento-e-

pesquisa/planejamento/evolucao-da-malha-rodoviaria/relatorio-sgp-2012-2013-brasil.pdf>. Acesso em 15 de setembro de 2019.

SADREHAGHIGHI, Ideen. Essentials of CFD. [S. l.], p. 1–465, 2019.

SCHMITZ, M. I. **Determinação de parâmetros termodinâmicos para projetos de equipamentos de refrigeração para carrocerias frigoríficas**. 2016. CENTRO UNIVERSITÁRIO UNIVATES, 2016.

SENGUTTUVAN, S. et al. Enhanced airflow in a refrigerated container by improving the refrigeration unit design. **International Journal of Refrigeration**, v. 120, p. 460–473, 2020. Disponível em: <https://doi.org/10.1016/j.ijrefrig.2020.08.019>.

SEO, D. W.; OH, J.; JANG, J. Performance analysis of a horn-type rudder implementing the Coanda effect. **International Journal of Naval Architecture and Ocean Engineering**, v. 9, n. 2, p. 177–184, 2017. Disponível em: <http://dx.doi.org/10.1016/j.ijnaoe.2016.09.003>.

SILVA, D. O. **Otimização da Separação Sólido-Líquido em Hidrociclones Mediante Modificações Geométricas**. 2012. Universidade Federal de Uberlândia, 2012.

SPAGNOL, W. A.; SILVEIRA, V.; PEREIRA, E.; FILHO, N. G. Monitoramento da cadeia do frio: Novas tecnologias e recentes avanços. Brazilian Journal of Food Technology, v. 21, p. 2–8, 2018.

UMENO, Y. et al. The use of CFD to simulate temperature distribution in refrigerated containers. **Engineering in Agriculture, Environment and Food**, v. 8, n. 4, p. 257–263, 2015. Disponível em: <http://dx.doi.org/10.1016/j.eaef.2015.03.002>.

VERMA, S.; SINGH, Harjit. Vacuum insulation in cold chain equipment: A review. **Energy Procedia**, *[S. l.]*, v. 161, p. 232–241, 2019. DOI: 10.1016/j.egypro.2019.02.086. Disponível em: <https://doi.org/10.1016/j.egypro.2019.02.086>.

VERMA, S.; SINGH, Harjit. Vacuum insulation panels for refrigerators. **International Journal of Refrigeration**, *[S. l.]*, v. 112, p. 215–228, 2020. DOI: 10.1016/j.ijrefrig.2019.12.007. Disponível em: <https://doi.org/10.1016/j.ijrefrig.2019.12.007>.

VERSTEEG, H. K.; MALALASEKERA, W. **An Introduction to Computacional Fluid Dynamics: The finite volume method**. 2. ed. England: Pearson Education Limited, 2007.

YILDIZ, T. CFD **Characteristics of Refrigerated Trailers and Improvement of Airflow for Preserving Perishable Foods**. Logistics, v. 3, n. 2, p. 11, 2019.

ZHAO, Y.; ZHANG, X.; XU, X. **Application and research progress of cold storage technology in cold chain transportation and distribution**. Journal of Thermal Analysis and Calorimetry, v. 139, n. 2, p. 1419–1434, 2020. Disponível em: <https://doi.org/10.1007/s10973-019-08400-8>.

ZHAO, C. J. et al. A review of computational fluid dynamics for forced-air cooling process. **Applied Energy**, v. 168, p. 314–331, 2016. Disponível em: <http://dx.doi.org/10.1016/j.apenergy.2016.01.101>.

ZHAO, Y.; ZHANG, X.; XU, X. **Application and research progress of cold storage technology in cold chain transportation and distribution**. Journal of Thermal Analysis and Calorimetry, v. 139, n. 2, p. 1419–1434, 2020. Disponível em: <https://doi.org/10.1007/s10973-019-08400-8>.

ZHU, Y. F.; XIE, J. Simulation and experiment of temperature field of different refrigerated trucks. **IOP Conference Series: Earth and Environmental Science**, v. 594, n. 1, 2020.